AD-A158438

AFWAL-TR-84-3035

FINITE ELEMENT ANALYSIS OF SOFT-BODY IMPACT

Robert A. Brockman

University of Dayton Research Institute
300 College Park Avenue
Dayton, Ohio 45469

October 1984

FINAL REPORT FOR PERIOD SEPTEMBER 1982 - MARCH 1984

FLIGHT DYNAMICS LABORATORY
AIR FORCE WRIGHT AERONAUTICAL LABORATORIES
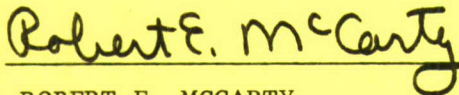AIR FORCE SYSTEMS COMMAND
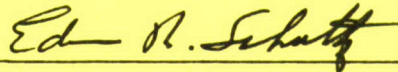WRIGHT-PATTERSON AIR FORCE BASE, OHIO 45433

NOTICE

When Government drawings, specifications, or other data are used for any purpose other than in connection with a definitely related Government procurement operation, the United States Government thereby incurs no responsibility nor any obligation whatsoever; and the fact that the government may have formulated, furnished, or in any way supplied the said drawings, specifications, or other data, is not to be regarded by implication or otherwise as in any manner licensing the holder or any other person or corporation, or conveying any rights or permission to manufacture, use, or sell any patented invention that may in any way be related thereto.

This report has been reviewed by the Office of Public Affairs (ASD/PA) and is releasable to the National Technical Information Service (NTIS). At NTIS, it will be available to the general public, including foreign nations.

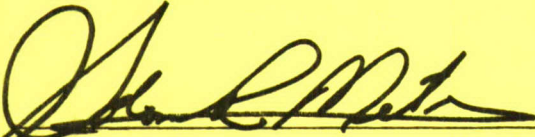This technical report has been reviewed and is approved for publication.

ROBERT E. MCCARTY
Project Engineer

EDWIN R. SCHULTZ, Chief
Crew Escape & Subsystems Branch

FOR THE COMMANDER

SOLOMON R. METRES, Chief
Vehicle Equipment Division

REPORT DOCUMENTATION PAGE

1a. REPORT SECURITY CLASSIFICATION UNCLASSIFIED	1b. RESTRICTIVE MARKINGS

2a. SECURITY CLASSIFICATION AUTHORITY	3. DISTRIBUTION/AVAILABILITY OF REPORT
2b. DECLASSIFICATION/DOWNGRADING SCHEDULE	Approved for public release; distribution unlimited.

4. PERFORMING ORGANIZATION REPORT NUMBER(S) UDR-TR-84-09	5. MONITORING ORGANIZATION REPORT NUMBER(S) AFWAL-TR-84-3035

6a. NAME OF PERFORMING ORGANIZATION University of Dayton Research Institute	6b. OFFICE SYMBOL *(If applicable)*	7a. NAME OF MONITORING ORGANIZATION Flight Dynamics Laboratory AF Wright Aeronautical Laboratories
6c. ADDRESS *(City, State and ZIP Code)* 300 College Park Avenue Dayton, Ohio 45469		7b. ADDRESS *(City, State and ZIP Code)* Air Force Systems Command Wright-Patterson Air Force Base, OH 45433

8a. NAME OF FUNDING/SPONSORING ORGANIZATION	8b. OFFICE SYMBOL *(If applicable)*	9. PROCUREMENT INSTRUMENT IDENTIFICATION NUMBER F33615-82-K-3408

8c. ADDRESS *(City, State and ZIP Code)*		10. SOURCE OF FUNDING NOS.			

	PROGRAM ELEMENT NO.	PROJECT NO.	TASK NO.	WORK UNIT NO.
11. TITLE *(Include Security Classification)* FINITE ELEMENT ANALYSIS OF SOFT-BODY IMPACT	62201F	2402	240203	24020344

12. PERSONAL AUTHOR(S)
Brockman, Robert A.

13a. TYPE OF REPORT Final	13b. TIME COVERED FROM 9/82 TO 3/84	14. DATE OF REPORT *(Yr., Mo., Day)* October 1984	15. PAGE COUNT 116

16. SUPPLEMENTARY NOTATION

17.	COSATI CODES		18. SUBJECT TERMS *(Continue on reverse if necessary and identify by block number)*
FIELD	GROUP	SUB. GR.	Nonlinear Structural Analysis Hydrodynamic Impact Finite Element Method Aircraft Transparency Soft Body Impact MAGNA Impact Loading

19. ABSTRACT *(Continue on reverse if necessary and identify by block number)*

This report describes the development of analytical and numerical methods for determining the dynamic structural response caused by soft body impact. The prediction of stress and deflection behavior of aircraft transparencies due to collisions with birds is the primary area of application. When the structure is relatively flexible, such an impact involves strong coupling between the dynamic motion of the colliding bodies. The developed techniques use appropriate finite element methods in each region, together with a contact solution algorithm, to compute the coupled structural response, while the soft-body motions are integrated using a simpler explicit technique. The soft body dynamic analysis uses a Lagrangian description to track the free surface and impact region accurately, with an automatic rezoning procedure to correct large mesh distortions.

20. DISTRIBUTION/AVAILABILITY OF ABSTRACT UNCLASSIFIED/UNLIMITED ☐ SAME AS RPT. ☒ DTIC USERS ☐	21. ABSTRACT SECURITY CLASSIFICATION UNCLASSIFIED

22a. NAME OF RESPONSIBLE INDIVIDUAL ROBERT E. MCCARTY	22b. TELEPHONE NUMBER *(Include Area Code)* (513) 255-2516	22c. OFFICE SYMBOL AFWAL/FIER

DD FORM 1473, 83 APR   EDITION OF 1 JAN 73 IS OBSOLETE.   UNCLASSIFIED

FOREWORD

This report describes the development of analytical methods for the analysis of soft-body impact, including the effects of structural response upon impact loading. The effort was conducted between September, 1982 and August, 1984, by the Analytical Mechanics Group (Dr. Ronald F. Taylor, Group Leader) of the Aerospace Mechanics Division (Mr. Dale H. Whitford, Supervisor) of the University of Dayton Research Institute, under Air Force Contract F33615-82-K-3408. The work described herein was performed under Project 2402, "Vehicle Equipment Technology," Task 240203, "Aerospace Vehicle Recovery and Escape Subsystems," Work Unit 24020344, "Coupled Structural Response of Aircraft Transparencies Caused by Soft-Body Impact."

Overall supervision of the project was provided by Mr. Dale H. Whitford. Dr. Fred K. Bogner was the original UDRI program manager, who was succeeded by Dr. Ronald F. Taylor. Dr. Robert A. Brockman was the principal investigator. Technical direction and support was provided by Mr. Robert E. McCarty (AFWAL/FIER) as the Air Force Project Engineer. The author also wishes to acknowledge the contribution of Mr. Vincent G. Dominic, whose assistance in developing and debugging the rezoning and coupled solution modules was invaluable.

TABLE OF CONTENTS

LIST OF ILLUSTRATIONS

LIST OF TABLES

SECTION 1
INTRODUCTION

This report describes the development of theoretical and numerical methods for use in simulating the dynamic response of structures to hydrodynamic loading. The particular problem of interest is that of soft-body impacts on aircraft transparencies, which involves several characteristics which make effective analysis quite difficult. Among these characteristics are

- nonlinear structural response (large displacements and nonlinear material behavior);

- structural failures which often occur at times far in excess of the duration of the impact;

- very large deformation of the impacting body;

- strong interaction between structural response and the magnitude and distribution of impact pressures; and

- free-surface boundary conditions on the impacting body.

The analytical techniques presented here address each of these problem areas to provide effective numerical solutions to practical problems involving soft body impact.

The remainder of this Section describes the technical aspects of the soft-body impact problem in more detail, and outlines the general approach adopted in its solution. Subsequent Sections are devoted to presenting the theoretical development and essential information about the computer implementation of the methodology.

1.1 PROBLEM OVERVIEW

Contact and impact situations are among the most important and difficult problems in computational mechanics. Contact or impact between solid bodies occurs in forming operations, wheel-rail contact, ballistic impacts, and numerous other practical circumstances. Fluid-structure interaction, contact between

solid and fluid media, is of interest in submerged structures, reactor vessels, and turbomachinery. For each of these classes of analytical problems, solution procedures of many types have been developed and used successfully [1-5]. For the case of solid to solid contact, very general analysis techniques exist, based upon both explicit and implicit integration methods. In fluid-structure interaction problems, the solution procedure necessarily reflects the specific problem geometry or response characteristics rather strongly.

Soft-body impacts, such as bird impacts on aircraft transparencies, involve the interaction of a small volume of solid material which behaves hydrodynamically (similar to a fluid) with a more massive structure whose response is generally less severe. When Eulerian methods of description (based on a mesh fixed in space) are used for the hydrodynamic solution, tracking of boundaries and free surfaces is difficult. Lagrangian mesh techniques, in which computational reference points move with the material, fail due to the severe mesh distortion which may occur within the soft body. Explicit, large deformation codes for solid mechanics applications [6-8] sometimes provide the type of analytical capabilities which are necessary to predict the resulting behavior; however, the need to compute structural response over relatively long times (several milliseconds) makes the use of these techniques prohibitive in terms of cost. Due to the lack of solution methodology which is truly tailored to the soft-body impact problem, ad-hoc techniques abound [9-11]. While such methods are economical and provide useful data for preliminary design, their reliability leaves much to be desired, and the objective of replacing experiment by analysis remains unfulfilled.

2

1.2 GENERAL APPROACH

The solution techniques presented here are based upon the finite element method [12]. Due to the widely differing characteristics of the soft body and the structure, the coupled impact problem is solved separately (but in tandem) for each region, with appropriate interface conditions being imposed at each stage of the solution.

The structural model experiences relatively small deformations, which must be computed for extended time periods in some cases. For the most part, structural response (as opposed to wave propagation) is of primary interest. Therefore, the finite element solution for the structure motion uses higher-order elements in conjunction with highly stable, implicit integration techniques. This combination limits the model to reasonable size and permits the use of time steps in the millisecond range. In return for this convenience, the implicit method requires an iterative solution at each time step, and formation of a global stiffness matrix. This approach to the dynamic response solution is common in general-purpose finite element packages, such as ABAQUS [13], ADINA [14], ANSYS [15], MAGNA [16] and MARC [17].

The behavior of the soft body region is quite different in character from the structural response. Here the deformations are large and, due to the low material stiffness, depend upon wave propagation effects as well as overall deformation modes. The response is essentially hydrodynamic; that is, pressure levels are typically many times greater than the strength of the material. For this region, the numerical solution is based upon explicit techniques which permit a much more detailed treatment of the problem physics. Due to the free surface which makes up most of the boundary of the body, a Lagrangian mesh, which moves with the material, is used. The severe mesh distortion which occurs in some situations is resolved by occasional "rezoning"; that is, complete regeneration of the mesh. While mesh rezoning is often a source of difficulty in large deformation analyses,

3

the present technique includes automatic rezoning operations which are invoked when preset tolerances are exceeded, and do not require stopping and restarting the solution.

Appropriate interface conditions between the structure and soft body meshes consist of kinematic conditions (compatibility of displacements and velocities) and force/impulse conditions (interface pressures and impulses). In the structural model, forces exerted by the hydrodynamic mesh are applied directly to compute incremental displacements; for the soft body, imposed displacement conditions are provided by the structure model, and interface forces develop as natural boundary conditions. For the contact/impact calculations, the algorithm presented in Reference [18] is used. Since stable time steps for the hydrodynamic mesh are much smaller than for the structure mesh, a partitioning method [19] is needed to enforce the proper flow of information between the two distinct portions of the mesh.

The primary development areas in the work described are the hydrodynamic solution and the interface techniques. These are suitable for integration into most implicit, nonlinear structural analysis programs. In the present effort, we have chosen one particular program, MAGNA (Materially And Geometrically Nonlinear Analysis) in which to implement the coupled impact solution.

4

1.3 NOTATION CONVENTIONS

The notation used in the body of this report conforms to the following rules and conventions:

- Einstein's summation convention applies unless otherwise noted; that is, repeated indices imply summation over the range of the index;

- lower case subscripts are used for reference to spatial dimensions and have a range of three;

- upper case subscripts refer to the nodes of a finite element, and therefore have a range of N for an N-noded element;

- material time derivatives are indicated by a superimposed dot, ($\cdot$);

- a comma represents partial differentiation with respect to the spatial coordinate which follows (e.g., $u_{,i} = \partial u / \partial x_i$); where both time and spatial derivatives occur, as in $\dot{u}_{,i}$, the material time derivative is evaluated first.

SECTION 2
THEORETICAL FORMULATION

In this Section, the theory and numerical algorithms used in the structural/hydrodynamic impact analysis are described. Since the structural dynamics solution relies heavily upon existing software, the discussion of this portion of the solution is only cursory; for further details, we encourage the reader to consult the documentation for one or more of the structural analysis codes mentioned in Subsection 1.2. Herein, emphasis is placed upon the hydrodynamic solution, the finite elements used, and the mesh rezoning algorithm.

2.1 EQUATIONS OF MOTION

In what follows, a general formulation of the soft-body impact problem is developed. Equations of motion are expressed in terms of convected coordinates which coincide with initial or reference positions of a body.

Consider two unconnected domains V_a and V_b, whose union is denoted V, with boundary ∂V (Figure 2.1). The boundary consists of three segments: ∂V_σ, on which external forces are prescribed, ∂V_u, on which displacements are specified, and $\partial V_c = \partial V_a \cap \partial V_b$, over which the two bodies are in contact. The initial position of a material point in V is described in terms of Cartesian coordinates a_m; $m = 1, 2, 3$, and its position at time t is $x_i(a_m, t)$; $i = 1, 2, 3$.

The motion of both bodies must obey the momentum equation

$$\sigma_{ji,j} + \rho \bar{f}_i = \rho \ddot{x}_i \qquad \text{in } V \tag{2.1}$$

the traction boundary conditions

$$\sigma_{ji} n_j = \bar{t}_i \qquad \text{on } \partial V_\sigma \tag{2.2}$$

the kinematic boundary conditions

$$u_i(a_m, t) = \bar{u}_i(t) \qquad \text{on } \partial V_u \tag{2.3}$$

the contact interface conditions

$$[\sigma_{ji}^{(a)} - \sigma_{ji}^{(b)}] n_j = 0 \tag{2.4}$$

$$[u_i^{(a)} - u_i^{(b)}] n_j = 0 \qquad \text{on } \partial V_c \tag{2.5}$$

and the initial conditions

$$x_i(a_m, 0) = a_i \tag{2.6}$$

$$\dot{x}_i(a_m, 0) = \bar{S}_i(a_m) \tag{2.7}$$

7
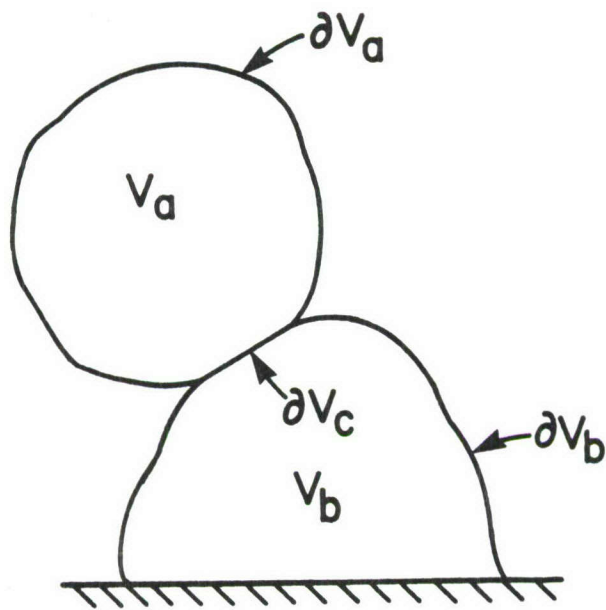
Figure 2.1. Contact Between Two Arbitrary Bodies.

8

In Equations (2.1)-(2.7), all force and stress measures refer to the instantaneous configuration, and overbars signify prescribed quantities.

We assume that all kinematic conditions (Equations 2.3 and 2.5) are satisfied identically. Equations (2.1), (2.2) and (2.4) may be expressed in the weak form

$$\int_V (\rho \ddot{x}_i - \sigma_{ji,j} - \rho \bar{f}_i) \delta x_i \, dV + \int_{\partial V_\sigma} (\sigma_{ji} n_j - \bar{t}_i) \delta x_i \, dA$$
$$+ \int_{\partial V_c} [\sigma_{ji}^{(a)} - \sigma_{ji}^{(b)}] n_j \, \delta x_i \, dA = 0 \qquad (2.8)$$

where δx_i are virtual displacements which satisfy the kinematic boundary conditions but are otherwise arbitrary. The divergence theorem gives the relation

$$\int_V (\sigma_{ji} \delta x_i)_{,j} dV = \int_{\partial V_\sigma} \sigma_{ji} n_j \delta x_i \, dA + \int_{\partial V_c} [\sigma_{ji}^{(a)} - \sigma_{ji}^{(b)}] n_j \delta x_i \, dA \qquad (2.9)$$

in which

$$\int_V (\sigma_{ji} \delta x_i)_{,j} dV = \int_V \sigma_{ji,j} \delta x_i \, dV + \int_V \sigma_{ji} \delta x_{i,j} \, dV$$

Equation (2.8) can therefore be rewritten in the more useful form

$$\int_V (\rho \ddot{x}_i \delta x_i + \sigma_{ji} \delta x_{i,j} - \rho \bar{f}_i \delta x_i) dV - \int_{\partial V_\sigma} \bar{t}_i \delta x_i \, dA = 0 \qquad (2.10)$$

The above weak form of the momentum equation and force boundary conditions, together with appropriate constitutive relationships, provides a basis for the finite element discretization of both material regions.

2.2 FINITE ELEMENT APPROXIMATION

The spatial discretization of Equation (2.10) is based upon a finite element approximation which, for a typical element "e", has the form

$$x_i^e(a_{m,t}) = \sum_{K=1}^{N_e} \phi_K^e(\xi, \eta, \zeta) \, x_{ij}^e(t) \qquad (2.11)$$

Here the upper case subscript refers to individual nodes of the element. The quantities $x_{ik}^e(t)$ are nodal values of the spatial coordinates, which are made continuous between adjacent elements using standard assembly procedures [12]. The shape functions $\phi_K^e(\xi, \eta, \zeta)$ are uniquely defined by the properties

$$\phi_K^e(\xi_I, \eta_I, \zeta_I) = \delta_{IK} \qquad \text{in } V_e \qquad (2.12)$$

$$\phi_K^e(\xi, \eta, \zeta) = 0 \qquad \text{outside } V_e$$

Here δ_{IK} is the Kronecker delta, which equals one when I and K are equal, and equals zero otherwise. In most cases, we can omit the superscript "e" without ambiguity, and simply write

$$x_i(a_{m,t}) = \phi_K x_{iK} \qquad (2.13)$$

Note that summation is implied on the repeated index K, whose range is equal to the number of nodes connected to an element.

The weak form of the equations of motion (Equation 2.10) becomes, in finite element form,

$$\sum_{e=1}^{E} \int_{V_e} [(\rho \ddot{x}_{iJ} \phi_j \phi_K \delta x_{iK} + \sigma_{ji} \phi_{K,j} \delta x_{iK} - \rho \bar{f}_i \phi_K \delta x_{iK}) \, dV \qquad (2.14)$$

$$- \int_{\partial V_\sigma^e} \bar{t}_i \, \phi_K \delta x_{iK} \, dA] = 0$$

Since the nodal virtual displacements δx_{iK} are arbitrary and independent of one another, their coefficients must vanish identically. This condition gives the semi-discrete equations of motion for the finite element model,

$$\sum_{e=1}^{E} \left[\int_{V_e} (\rho \ddot{x}_{iJ} \phi_j \phi_K + \sigma_{ji} \phi_{K,j} - \rho \bar{f}_i \phi_K) \, dV \right.$$

$$\left. - \int_{\partial V_\sigma^e} \bar{t}_i \phi_K \delta x_{iK} \, dA \right] = 0 \qquad (2.15)$$

Defining the element mass matrix

$$M_{JK}^e = \int_{V_e} \rho \phi_j \phi_K \, dV \qquad (2.16)$$

and the element force vector

$$F_{iK}^e = \int_{V_e} (\rho \bar{f}_i \phi_K - \sigma_{ji} \phi_{K,j} {}^K) \, dV + \int_{\partial V_\sigma^e} \bar{t}_i \phi_K \, dA \qquad (2.17)$$

Equation (2.15) becomes

$$\sum_{e=1}^{E} (M_{JK}^e \ddot{x}_{iJ} - F_{iK}^e) = 0 \qquad (2.18)$$

Finally, invoking the finite element assembly procedure gives the system of equations

$$M_{JK} \ddot{x}_{iJ} = F_{iK} \qquad (2.19)$$

in which the nodal (upper case) subscripts range from one to the number of nodes in the model. Equations (2.19), subject to the kinematic boundary conditions (Equations 2.3 and 2.5) and the initial conditions (Equations 2.6 and 2.7) must be integrated in time to determine the transient response of the finite element model.

11

2.3 STRUCTURAL RESPONSE SOLUTION

The structural responses of interest are dominated by low
frequency wave motion, and involve moderate amounts of material
deformation. Compared with these lower frequency motions, the
highest frequencies of the computational mesh for the structure
are many times greater due to high material stiffness. To
capture the overall structural motion using time steps which are
not controlled by high frequency behavior of the mesh, an
implicit method of solution is appropriate. Implicit solution
techniques make use of the equations of motion at time t+Δt to
solve for the motion at t+Δt, and normally involve the solution
of simultaneous equations and/or iteration at each time step.

As the basis of the implicit integration we adopt the two
parameter Newmark algorithm [20], which is obtained from the
following finite difference approximations:

$$\dot{x}_{iJ}^{t+\Delta t} = \dot{x}_{iJ}^{t} + [(1-\delta)\ddot{x}_{iJ}^{t} + \delta\ddot{x}_{iJ}^{t+\Delta t}] \qquad (2.20)$$

$$x_{iJ}^{t+\Delta t} = x_{iJ}^{t} + \dot{x}_{iJ}^{t}\Delta t + [(\tfrac{1}{2}-\alpha)\ddot{x}_{iJ}^{t} - \alpha\ddot{x}_{iJ}^{t+\Delta t}] \, (\Delta t)^2$$

Parameters α and δ are arbitrary and may be chosen to obtain
desirable numerical properties. The values $\alpha=1/4$, $\delta=1/2$ give the
trapezoidal integration rule, which is used exclusively in this
report.

In the implicit solution, Equation (2.19) is applied at time
t+Δt to determine the solution $x_{iJ}^{t+\Delta t}$:

$$M_{JK} \, \ddot{x}_{iJ}^{t+\Delta t} = F_{iK}^{t+\Delta t} \qquad (2.21)$$

Since the forces $F_{iK}^{t+\Delta t}$, which include the internal forces, depend
upon $x_{iJ}^{t+\Delta t}$ in a nonlinear fashion, Equation (2.21) must be solved

iteratively. We define the nodal residual forces

$$\Phi_{iK} = F_{iK}^{t+\Delta t} - M_{JK} \ddot{x}_{iJ}^{t+\Delta t} \qquad (2.22)$$

which must vanish at the true solution $\bar{x}_{iJ}^{t+\Delta t}$. Zeros of Equation (2.22) can be generated by Newton-Raphson iteration based upon the Taylor series approximation

$$\Phi_{iK}(\bar{x}_{MN}^{t+\Delta t}) = \Phi_{iK}(x_{MN}^{t+\Delta t}) + \frac{\partial \Phi_{iK}}{\partial x_{jL}} (\bar{x}_{jL}^{t+\Delta t} - x_{jL}^{t+\Delta t}) + \ldots = 0 \quad (2.23)$$

The Jacobian matrix which must be formed, factored and solved to carry out the iteration is otained by substituting the accelerations $\ddot{x}_{iJ}^{t+\Delta t}$ from Equations (2.20) into Equation (2.22), and differentiating with respect to x_{jL}. Notice that, since the quantities x_{iJ} are discrete parameters, we have $\partial x_{iJ}/\partial x_{jL} = \delta_{ij}\delta_{JL}$. The result is:

$$\frac{\partial \Phi_{iK}}{\partial x_{jL}} = \frac{\partial F_{iK}}{\partial x_{jL}} + \frac{1}{\alpha(\Delta t)^2} M_{LK}\delta_{ij} \qquad (2.24)$$

where δ_{ij} is the Kronecker delta. The leading term in Equation (2.24) is commonly called the tangent stiffness matrix.

Equation (2.23) is applied repeatedly at each step of the dynamic solution, until the corrections $\bar{x}_{jL}^{t+\Delta t} - x_{jL}^{t+\Delta t}$ and/or the residuals Φ_{iK} are smaller than predefined tolerances. Once the displacement solution is known to the prescribed accuracy, nodal velocities and accelerations are updated using Equations (2.20). Reference [16] describes the implicit solution for the structural response in more detail.

13

2.4 HYDRODYNAMIC RESPONSE SOLUTION

In contrast to the structural motion, the impact behavior of the soft body is hydrodynamic in nature and involves very large material distortions. Furthermore, wave speeds in this portion of the computational mesh are relatively small due to low material stiffness. Here a simpler method of solution is most appropriate.

Integration of the equation of motion in the hydrodynamic mesh is performed using the central difference approximations

$$\dot{x}_{iJ}^{t+\Delta t/2} = \frac{1}{\Delta t} (x_{iJ}^{t+\Delta t} - x_{iJ}^{t})$$

$$\ddot{x}_{iJ}^{t} = \frac{1}{\Delta t} (\dot{x}_{iJ}^{t+\Delta t/2} - \dot{x}_{iJ}^{t-\Delta t/2}) \qquad (2.25)$$

The solution in this case is explicit, since the state at time $t+\Delta t$ is determined directly from that at time t. Equations (2.19) and (2.25) are used directly, once per time step, to advance the acceleration, velocity and displacement solutions.

The explicit nature of Equation (2.25) makes the integration conditionally stable (that is, stable only for a limited range of step sizes). The stability limit is well-known [21], and is given by

$$\Delta t_{cr} = \frac{2}{\omega_{max}} [\sqrt{1-\epsilon^2} - \epsilon] \qquad (2.26)$$

where ω_{max} is the highest frequency of the mesh, and ϵ is the fraction of critical damping in the highest frequency. Since it is inconvenient to estimate the maximum frequency of the mesh directly, we note that the maximum frequency for the complete mesh is bounded above by the maximum element frequency. It is therefore possible to relate the highest frequency to the nodal separation $\Delta\ell$ and the material sound speed c,

14

$$\omega_{max} = \left(\frac{2c}{\Delta \ell}\right)_{max} \qquad (2.27)$$

Equation (2.27) can be applied element-by-element at each time step to control the integration time step. The acoustic speed c for an inviscid flow is simply

$$c = \left.\frac{\partial p}{\partial \rho}\right)_s \qquad (2.28)$$

where the symbol s represents the entropy. This quantity can be obtained for each element as a by-product of the equation of state calculations.

2.5 HYDRODYNAMIC FINITE ELEMENTS

The details of the specific finite element approximation adopted for the hydrodynamic solution are described in this section. Basic aspects of the element formulation follow the work of Flanagan and Belytschko [21].

2.5.1 Element Geometry and Shape Functions

The unit of approximation in the hydrodynamic mesh is the eight-node, linear displacement hexahedron shown in Figure 2.2. A unit cube in the parametric coordinates (ξ, η, ζ), also denoted by ξ_i, is mapped into physical coordinates x_i by means of the shape functions $\phi_I(\xi, \eta, \zeta)$. If the center of the cube is taken to be the origin, $-1/2 \leq \xi_i \leq 1/2$, the shape functions are

$$\phi_I(\xi, \eta, \zeta) = (\tfrac{1}{2} + 2\xi_I \xi)(\tfrac{1}{2} + 2\eta_I \eta)(\tfrac{1}{2} + 2\zeta_I \zeta) \tag{2.29}$$

Flanagan and Belytschko [21] have shown that the ϕ_I also may be expressed in terms of an orthogonal set of base vectors

$$\phi_I = \tfrac{1}{8}\Sigma_I + \tfrac{1}{4}\xi\Lambda_{1I} + \tfrac{1}{4}\eta\Lambda_{2I} + \tfrac{1}{4}\zeta\Lambda_{3I}$$

$$+ \tfrac{1}{2}\eta\zeta\Gamma_{1I} + \tfrac{1}{2}\zeta\xi\Gamma_{2I} + \tfrac{1}{2}\xi\eta\Gamma_{3I} + \xi\eta\zeta\Gamma_{4I} \tag{2.30}$$

The constants Σ_I, Λ_{jI}, and $\Gamma_{\alpha I}$ are summarized in Table 2.1, and are hereafter referred to as the constant, linear, and hourglass base vectors, respectively. This form of the shape functions is useful since it isolates the contribution of hourglass modes of deformation (which are neglected by one-point integration) explicitly on the unit cube.

2.5.2 Mean Stress Approximation

The eight-node hexahedron, one of the simplest of three dimensional elements, must be used with care if good numerical behavior is to be obtained. An exact numerical integration of

16
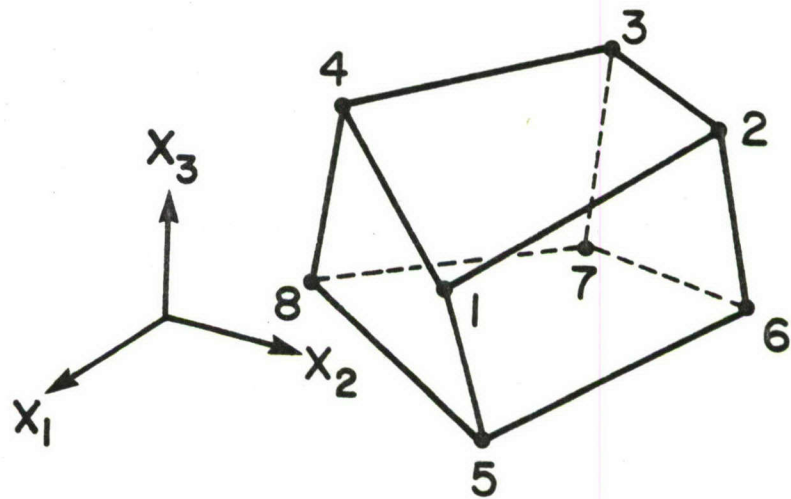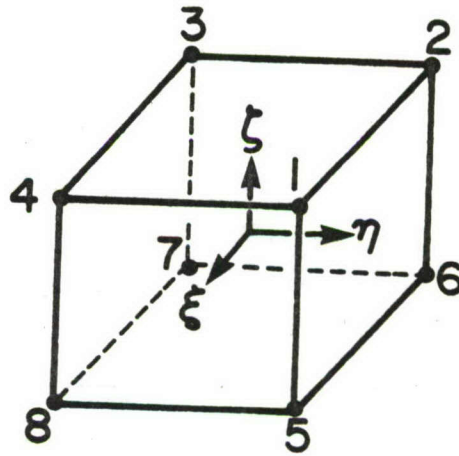
Figure 2.2.   Eight-Node Hexahedral Finite Element.

TABLE 2.1

BASIS VECTORS FOR EIGHT-NODE HEXAHEDRON

Node	ξ	η	ζ	Σ_I	Λ_{1I}	Λ_{2I}	Λ_{3I}	Γ_{1I}	Γ_{2I}	Γ_{3I}	Γ_{4I}
1	1/2	1/2	1/2	1	1	1	1	1	1	1	1
2	-1/2	1/2	1/2	1	-1	1	1	-1	1	-1	-1
3	-1/2	-1/2	1/2	1	-1	-1	1	1	-1	-1	1
4	1/2	-1/2	1/2	1	1	-1	1	-1	-1	1	-1
5	1/2	1/2	-1/2	1	1	1	-1	1	-1	-1	-1
6	-1/2	1/2	-1/2	1	-1	1	-1	-1	-1	1	1
7	-1/2	-1/2	-1/2	1	-1	-1	-1	1	1	1	-1
8	1/2	-1/2	-1/2	1	1	-1	-1	-1	1	-1	1

the eight node element may lead to "locking" of the mesh, while a
simple reduced (one point) integration fails to integrate the
volume of irregular elements exactly and may be non-convergent.
In the present work, we adopt the "mean stress approximation"
described in Reference [21] to obtain a convergent element with
desirable numerical properties.

The mean stress approximation is obtained by using mean
values of the stress and kinematic variables, while performing
the volume integration exactly for arbitrarily-shaped elements.
The resulting element is therefore capable of representing states
of uniform stress and deformation exactly, a property which is
necessary for convergence [12]. The sole disadvantage is that
hourglass deformation modes of the element may be unrestrained,
and recourse must be made to artificial damping mechanisms to
avoid contamination of the numerical solution.

2.5.3 Element Kinematics

The spatial coordinates of points within a single element
are approximated as indicated in Equation (2.11),

$$x_i = \phi_K x_{iK} \tag{2.31}$$

where the shape functions for the eight node hexahedron are
defined by Equation (2.30). The range of the nodal (upper case)
subscript is 8, the number of nodes per element. The velocity
field is expressed in a similar form

$$v_i = \phi_K v_{iK} \tag{2.32}$$

For the spatial velocity gradient, we obtain

$$v_{i,j} = \phi_{K,j} v_{iK} \tag{2.33}$$

since only ϕ_K varies with spatial position. The parameters v_{iK}

are nodal quantities which vary only with time, and hence $v_{iK,j} = 0$.

With the mean stress approximation, it is necessary to form mean values of the velocity gradient and stress for an element. For this purpose, it is convenient to define a matrix containing the integrals of the shape function derivatives,

$$B_{iJ} = \int_{V_e} \phi_{J,i} \; dV \tag{2.34}$$

For lack of a better name, matrix $\underset{\sim}{B}$ will be referred to as the "geometric matrix" for an element, since it is purely a function of the element nodal coordinates. Consider the mean velocity gradient for an element,

$$\bar{v}_{i,j} = \frac{1}{V_e} \int_{V_e} v_{i,j} \; dV \tag{2.35}$$

From Equation (2.34), $\bar{v}_{i,j}$ can be written more concisely as

$$\bar{v}_{i,j} = \frac{1}{V} B_{jK} v_{iK} \tag{2.36}$$

The geometric matrix $\underset{\sim}{B}$ also provides a convenient method for evaluating the element volume, since [21]

$$x_{iK} B_{jK} = V \delta_{ij} \tag{2.37}$$

where δ_{ij} is the Kronecker delta. It is straightforward to verify that both the constant and the hourglass base vectors are orthogonal to $\underset{\sim}{B}$,

$$B_{jI} \Sigma_I = 0 \qquad ; \; j=1,2,3 \tag{2.38}$$

$$B_{jI} \Gamma_{\alpha I} = 0 \qquad ; \; j=1,2,3; \; \alpha=1,2,3,4 \tag{2.39}$$

2.5.4 Constitutive Model

The constitutive relations used for the hydrodynamic mesh are relatively simple, since the larger-scale effects of impact (such as pressure level and momentum transfer) are of primary interest. The material is defined in terms of bulk and shear moduli (K,G), density ρ, and a material strength σ_y. The rate of deformation and the co-rotational (Jaumann) rate of the Cauchy stress are assumed to be related by

$$\overset{\triangledown}{\sigma}_{ij} = (K - \tfrac{2}{3}G)\, d_{kk}\delta_{ij} + 2G\, d_{ij} \tag{2.40}$$

where the Jaumann stress rate $\overset{\triangledown}{\sigma}_{ij}$ is defined by

$$\overset{\triangledown}{\sigma}_{ij} = \overset{\circ}{\sigma}_{ij} + \sigma_{iK}w_{kj} + \sigma_{jk}w_{ki} \tag{2.41}$$

In Equation (2.41), σ_{ij} is the stress tensor, $\overset{\circ}{\sigma}_{ij}$ its material time derivative, and w_{ij} is the spin (vorticity),

$$w_{ij} = \tfrac{1}{2}(v_{i,j} - v_{j,i}) = v_{i,j} - d_{ij} \tag{2.42}$$

Material strength is accounted for by computing an effective deviatoric stress,

$$\sigma_e = \sqrt{\tfrac{3}{2}\, \sigma'_{ij}\sigma'_{ij}} \tag{2.43}$$

which is limited in magnitude to the material strength σ_y. When the strength is exceeded, the deviatoric stresses are scaled according to

$$\sigma'_{ij} \leftarrow \frac{\sigma_y}{\sigma_e}\, \sigma'_{ij} \tag{2.44}$$

It should be noted that the primary purpose of the viscous terms in the constitutive model is to stabilize pure shear modes which are otherwise unconstrained. For this purpose, a small positive shear coefficient is normally used.

2.5.5 Element Forces

The element forces (Equation 2.19) include three separate contributions:

- prescribed external forces;
- internal forces; and
- hourglass-stabilizing forces.

The first two of these are physically meaningful. The hourglass resisting forces exist solely to stabilize the computational mesh and must not interfere with the representation of the problem physics.

Physical forces for an element are obtained from Equation (2.17). For simplicity, we will consider only body forces and the element internal forces,

$$F_{ik} = \int_V (\rho \bar{f}_i \phi_K - \sigma_{ji} \phi_{K,j})\ dV \tag{2.45}$$

Consistent with the mean stress approximation, ρ, $\bar{f}_i$, and σ_{ji} are replaced by their mean values over an element; the resulting nodal forces are then

$$F_{iK} = \frac{1}{8}\ \rho V \bar{f}_i \Sigma_K - \bar{\sigma}_{ji} B_{jK} \tag{2.46}$$

The mean stress $\bar{\sigma}_{ji}$ is obtained directly from the constitutive model and the mean velocity gradient.

Hourglass-resisting forces for the hexahedron are computed using the method described by Flanagan and Belytschko [21]. Figure 2.3 shows the four independent displacement modes of an element corresponding to a single component of displacement; the purpose of the anti-hourglassing forces is to resist the development of the four hourglass displacement modes, which are strain-free under the mean stress approximation.

22
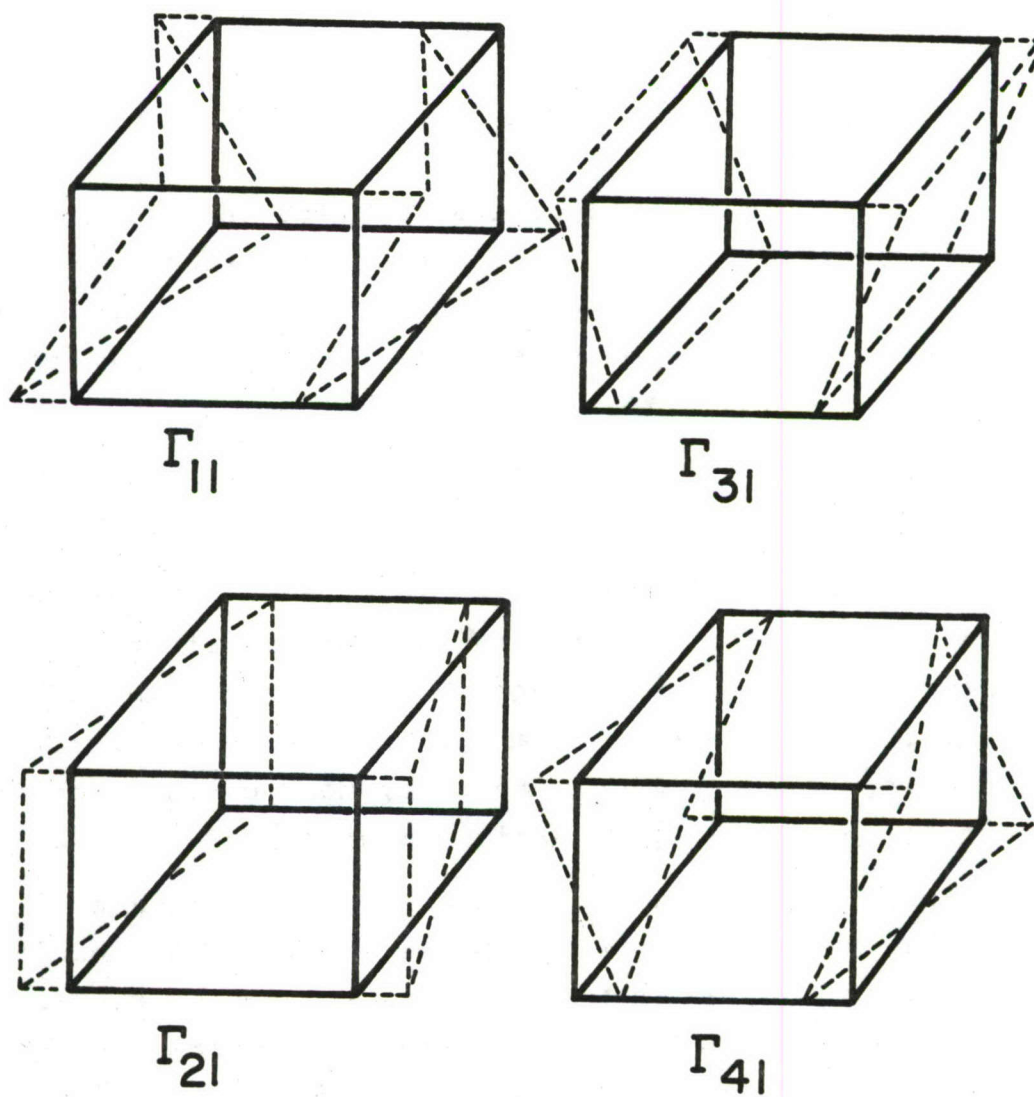
Figure 2.3. Hourglass Displacement Patterns for Eight-Node Hexahedron.

Define the hourglass shape vector $\gamma_{\alpha I}$ as

$$\gamma_{\alpha I} = \Gamma_{\alpha I} - \frac{1}{V} B_{iI} x_{iJ} \Gamma_{\alpha J} \qquad (2.47)$$

The first subscript refers to a particular hourglass displacement mode (Figure 2.3), and ranges from one to four; the upper case subscript refers to specific nodes of an element. The hourglass shape vector is orthogonal to the linearly varying portion of the element velocity field for an element of arbitrary shape. That is, the products

$$\dot{q}_{i\alpha} = \frac{1}{\sqrt{8}} v_{iK} \gamma_{\alpha K} \qquad (2.48)$$

called the hourglass modal velocities, provide an indication of the amount of hourglassing which is present in the element. The factor of $1/\sqrt{8}$ in Equation (2.48) is present to normalize the magnitudes of the modal velocities. Anti-hourglassing forces are proportional to the hourglass modal velocities,

$$F_{i\alpha}^{HG} = \epsilon \sqrt{\frac{\rho}{6}(K+\frac{4}{3}G)B_{jK}B_{jK}} \; \dot{q}_{i\alpha} \qquad (2.49)$$

and vanish when the element velocity field is purely linear. A damping coefficient ϵ is used to control the magnitude of the restoring forces. When the anti-hourglassing forces are defined as indicated in Equation (2.49), ϵ corresponds to the fraction of critical damping in the highest-frequency mode of the element, and critical time steps may be estimated using Equation (2.26).

It is important to realize that only pure hourglassing deformations are suppressed by the above scheme. When global deformation modes develop in which individual finite elements experience other than purely linear velocity fields, the anti-hourglassing forces combine to produce zero resultant forces, since the hourglass modal velocities in adjacent elements are identical in sign. In contrast to global deformation modes, true hourglassing patterns contain modal velocities which alternate in sign between adjacent elements, and the resistive forces combine to suppress such motion (Figure 2.4).

HOURGLASSING PATTERN

DEFORMATION PATTERN
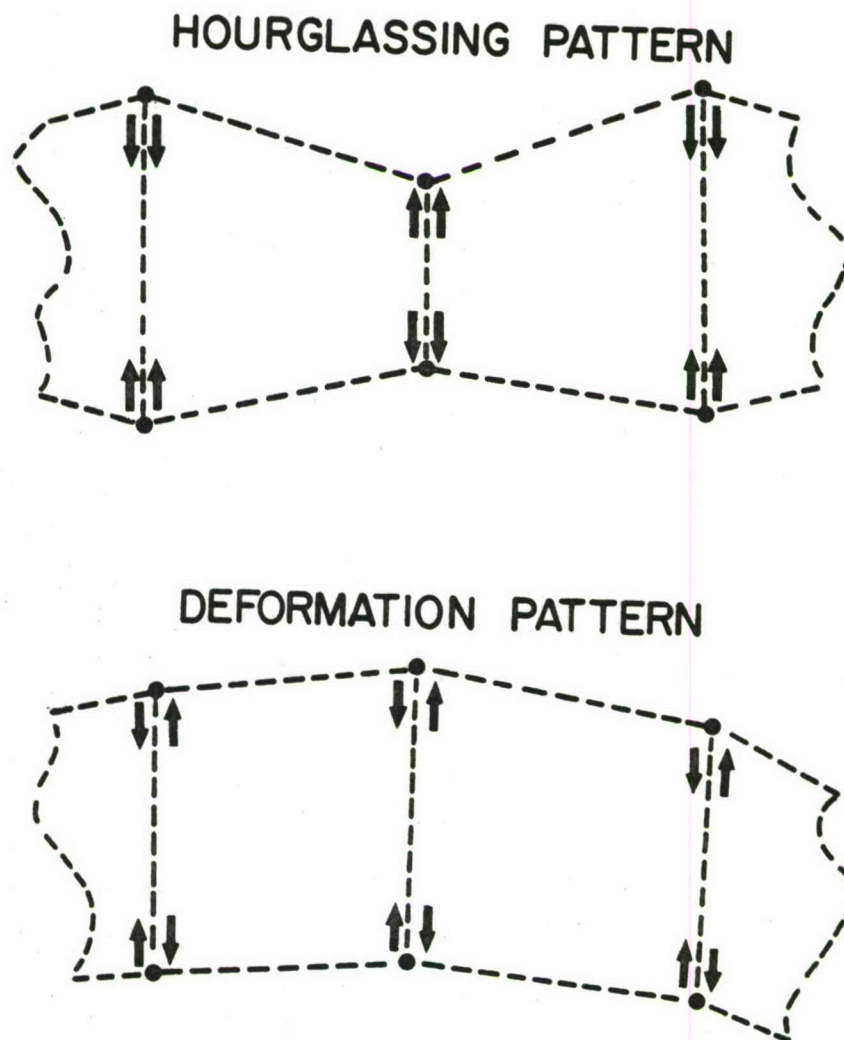
Figure 2.4.   Behavior of Hourglass-Resisting Forces in Adjacent Finite Elements.

2.6 MESH REZONING

In the hydrodynamic portion of the mesh, large deformations may occur as the solution proceeds. Since the nodal positions are updated at each time step, very large deformations may result in excessive distortion of the mesh, which are manifested in the following ways:

- disparities in element size due to volume change;
- singularities due to element volumes tending to zero;
- undesirable element shape due to large distortions; and
- reductions in the permissible time step due to extreme distortion and/or volume change.

These unfortunate situations can be resolved by reconstruction of the computational mesh (rezoning), performed as necessary during the solution.

Rezoning techniques are commonly used in large deformation problems of solid mechanics, such as ballistic impact and metal forming. However, their use normally requires a great deal of interaction with the analyst and is therefore time-consuming and expensive. The rezoning method developed in the present work is automatic and non-interactive, and therefore can be performed without halting the solution temporarily. In return for this convenience, our technique allows somewhat less control over the appearance of the rezoned computational mesh.

The rezoning process involves redefining nodal coordinates and element connections for the model, and relocating solution variables (velocities, pressures, densities, stresses) at the newly-defined nodal positions. In general, the size of the rezoned model (number of nodes and elements) may be different from that of the original model. Subsequent sections describe the conditions under which mesh rezoning is performed, and the main features of the rezoning algorithms used in the present work.

Two types of rezoning procedures have been developed and used to date; we refer to these as rezoning algorithms I and II in the

remainder of this section. The logic for invoking the rezoning process is identical for both algorithms, and is discussed in the next subsection.

2.6.1 Initiation of Rezoning Procedure

In most instances, severe distortion of the mesh can be detected by monitoring element volumes and computed critical time step sizes. If uniform compression or expansion occurs, critical time step sizes may change slowly while the elemental volumes change more rapidly. Severe shape distortion is usually accompanied by a rapid decrease in the allowable time increment.

The hydrodynamic solution module continuously monitors both the minimum and maximum element volumes for the entire mesh, as well as the allowable time step. Each time the hydrodynamic solution is restarted, initial values of these parameters are computed and stored. At each time step, current values are recovered from the internal force calculation, and changes in each quantity are compared with preset tolerances (user-specified or defaulted). When any one of the monitored values (minimum volume, maximum volume, critical time step), as compared with the initial values, exceeds its tolerance, rezoning is automatically invoked. Following the rezone, several key solution quantities (e.g., nodal masses) and the initial volume and time step values are reset, and the solution restarts from the point of interruption.

2.6.2 Rezoning Algorithm I

The first rezoning procedure consists of two phases: (a) the definition of new nodal points and the corresponding solution variables; and (b) the definition of finite elements which span the material volume. The algorithm is designed to produce a regular mesh within the interior of the region, with irregular elements used as necessary near the boundaries and free surfaces of the model.

The relocation of nodal points within the mesh is performed using a reference grid pattern specifying fixed mesh stations

27

along each of the Cartesian coordinate axes. Figure 2.5 shows such a reference grid pattern in two dimensions. Wherever material points coincide with intersections on this reference pattern, nodes will be positioned; on the boundaries of the region, nodes are located on reference lines as much as possible.

Figure 2.6 shows the relocation procedure in a conceptual form. All nodes within the interior of the body lie at reference grid intersections, and additional nodes (on reference lines if possible) describe the boundary of the region as required. As the positioning of new nodes takes place, linear interpolations within the existing finite elements are performed to define nodal values of the solution variables at the new node locations.

A key ingredient of the numerical algorithm used in rezoning is the establishment of a one to one correspondence between node points and mesh reference positions. When a conflict exists, it is resolved in favor of the (new) node which lies closest to the boundary of the material. This convention facilitates searching operations during nodal placement, the merging of boundary nodes, and the creation of finite elements for the rezoned mesh. In return for this convenience, the resolution of the procedure is limited to a linear interpolation between points which lie on adjacent reference lines.

This first phase of the rezoning process is accomplished by examining each element once, determining its intersections with the reference mesh pattern and then tentatively positioning new nodal points. Although new nodes may be moved several times as the rezone continues, this operation is quick since no searching is required. Once all elements have been searched, the boundary nodes are examined for excessively small spacing and merged when possible; this prevents critical time step sizes for the solution phase from becoming unreasonably small due to fine mesh spacing where none is called for.

Creation of new elements is straightforward once the nodal points have been redefined and merged. Each set of eight

28
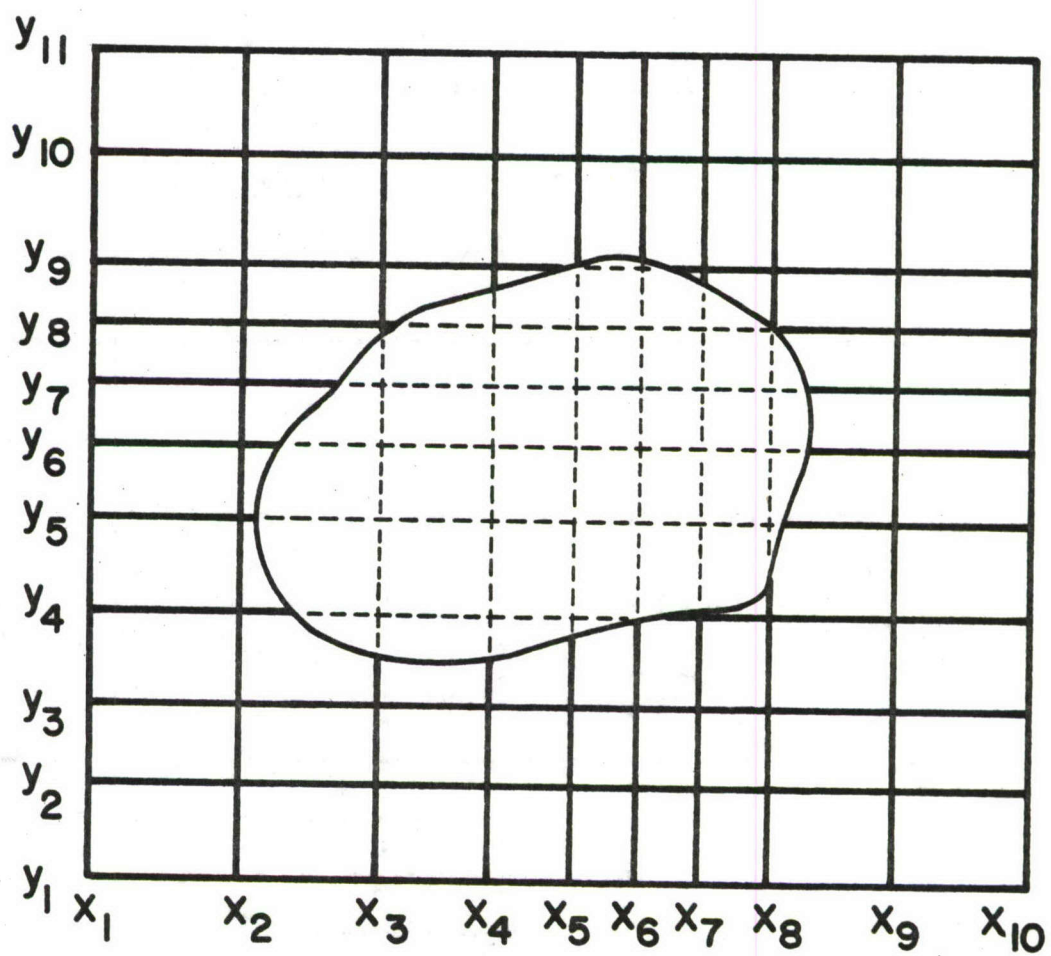
Figure 2.5. Typical Reference Grid for Mesh Rezoning.
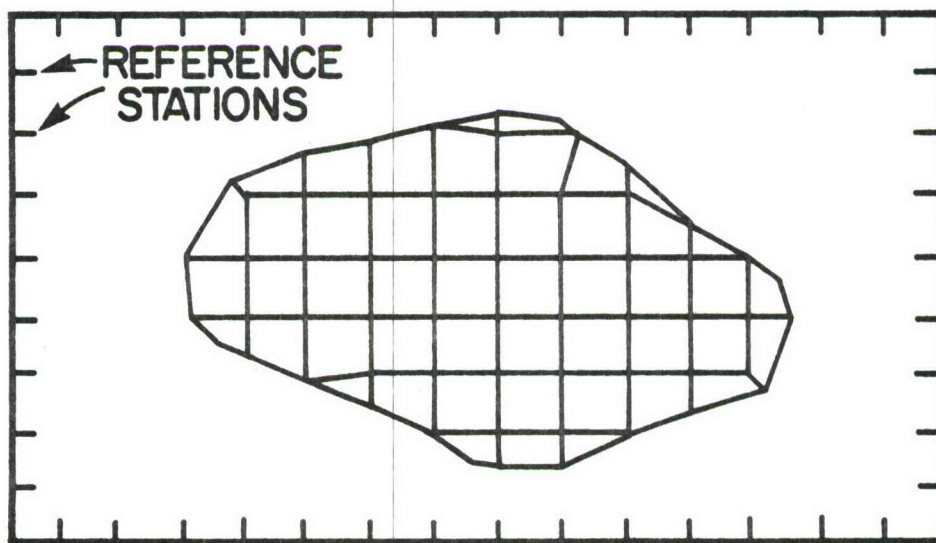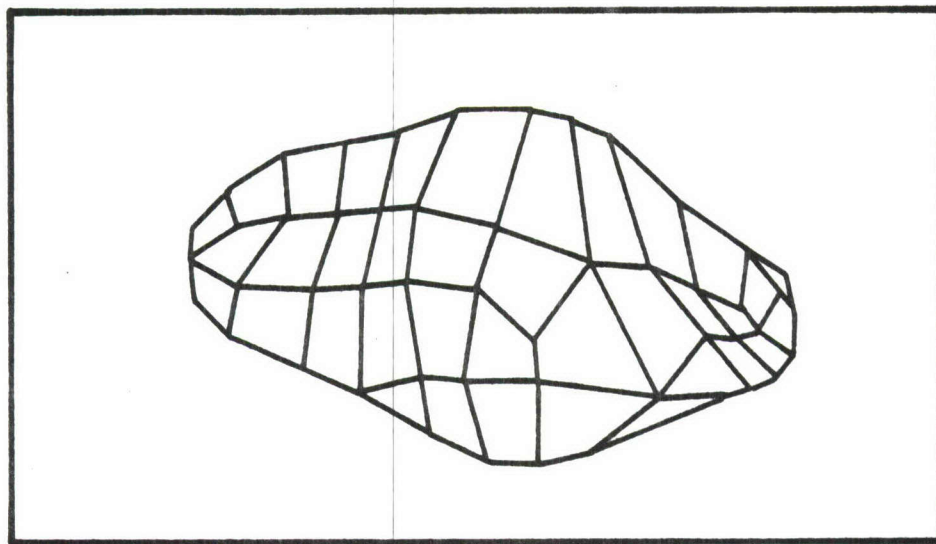
Figure 2.6.   Two-Dimensional Example of Rezoning Procedure.

adjacent reference positions potentially defines one element; to determine if an element is actually present, the nodes which correspond to each such group of points is examined. When at least four nodes are present, and the resulting element volume is positive, an element is defined.

2.6.3 Rezoning Algorithm II

A second rezoning technique has been developed based upon experiences gained in using Algorithm I. The function of this alternative technique is the same: given a distorted mesh of eight-node elements, generate a revised mesh in which the distortion is reduced or eliminated.

Algorithm II is based upon a different philosophy from Algorithm I, in that:

o the number of nodes and elements does not change;

o mass and momentum are conserved to machine accuracy; and

o the repositioning is less dramatic than with Algorithm I.

This second procedure is performed by examining each node of the existing mesh, and determining whether the node lies within the interior of the body, on a smooth surface, on a distinct "edge" of the region, or at a current "corner." The distinction between these cases is made by examining the element faces connected to each node.

For each point in the model, all element faces connected to the point are collected, and a geometric analysis much like a hidden line/surface removal procedure is performed. For an interior node, all connected surfaces are eliminated (thus identifying the node as interior). In this case, neighboring points are used to reposition the node in question, and solution variables are mapped to the new geometry by conditions of mass and momentum conservation, and by volume weighting of tensor-valued quantities (such as stress). A series of similar tests have been designed to isolate surface, edge, and corner nodes. The sole difference in treating these additional cases lies in the selection of surrounding nodes to be used in repositioning. For

surface nodes, only those nodes on the same surface are used, in order to maintain the surface shape; "edge" nodes are moved only along the curve defining the edge in question. Corner points are never relocated, since they represent locations at which the model geometry may be sharply discontinuous.

Our experience has been that the second rezoning algorithm is superior, since mass, momentum, and geometric shape are preserved. Symmetry conditions, when they exist, tend to survive many repeated rezones with Algorithm II as well. Finally, the second algorithm requires somewhat less memory and is faster in execution than Algorithm I.

The sole disadvantage to the alternative rezoning procedure is that nodal repositioning is moderate compared with the first algorithm, so that rezoning must be performed more frequently. The most satisfactory performance has been obtained with relatively mild tolerances (about 50%) for both maximum element volume change and time step reduction.

The soft-body impact code has been written to use either of the rezoning algorithms described in this section. However, Algorithm II is now being used exclusively due to its superior performance.

2.7   STRUCTURE MODEL / HYDRODYNAMIC MODEL INTERFACE

The interface between structural and hydrodynamic models uses the contact analysis technique described in References [16] and [18]. "Interface" or "contact" elements are defined on the surface of the structure model; these elements are strictly geometric, and have no material characteristics. The interface elements move with the structure model, and cannot be penetrated by nodes of the hydrodynamic model. Thus, the interface elements impose time-dependent displacement boundary conditions on the hydrodynamic model. Forces at hydrodynamic nodes in contact with the interface elements define the impact forces, which are transferred to the structural model at each major time step. At minor time steps (i.e., those in the soft-body mesh), only kinematic calculations (related to the displacement boundary conditions) are performed.

The interface elements used in the present work are defined by four corner points, as shown in Figure 2.7. Ordering of the nodal points determines the outward normal direction for an element. The restriction to four points rather than the variable number of nodes used in the original algorithm is for reasons of computational economy, since contact calculations must be performed at each time step for the hydrodynamic mesh.

The sole remaining differences between the interface element calculations in the present work and the original algorithm are related to the ordering of the computations. Since the interface element positions change only at major (structure) time steps, while contact searches must be done at minor (hydrodynamic) time steps, geometric parameters for the interface are precomputed at each major step and stored for repeated use. These parameters include element local coordinate transformations, coordinates, and limiting coordinate values.

33
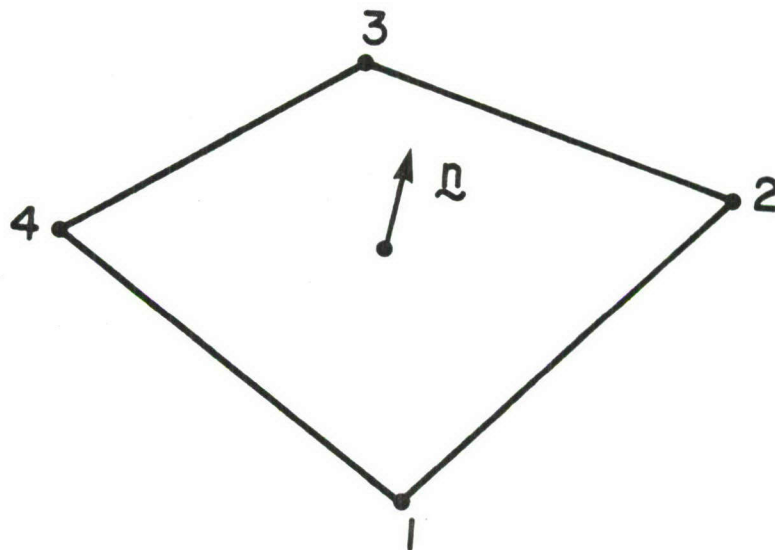
Figure 2.7.   Structural-Hydrodynamic Interface Element.

SECTION 3
DEMONSTRATION PROBLEMS

Testing of the fluid-structure impact analysis procedure has included the following types of problem solutions:

o elastic vibrations (using the moving boundary option);

o low-speed normal impacts on rigid surfaces;

o classical fluid flow cases, such as Couette flow (using a Newtonian fluid material model);

o oblique impacts on rigid surfaces; and

o coupled solutions for normal fluid-structure impact.

In most cases, these solutions have been used to verify the correctness of the soft-body impact analysis module. However, some cases (mostly oblique impacts) have been used only to study the behavior of the anti-hourglassing formulation and to verify the correct operation of the rezoning module for problems involving fully three-dimensional response. A fully coupled impact test case is described in detail below.

The coupled problem considered is the normal impact of a cylinder of porous gelatin, used in the laboratory simulation of bird impacts, on a square flat plate. Although experimental data corresponding precisely to the problem considered do not exist, a similar case of impact on a rigid target is reported in Reference [23]. The objectives in studying a similar case with a flexible target are to verify that the computed force levels are similar, and to observe the performance of the contact/impact procedure in a situation where kinematic constraints and interface forces can be observed and interpreted simply.

Physical data used in the simulation are as follows. The plate is 12 inches square and 0.1 inch thick; one quadrant of the plate is modeled due to symmetry. The plate material is aluminum, with a modulus of 10 Mpsi, Poisson's ratio 0.3, and density 0.098 pounds per cubic inch. All of the outer boundaries of the plate are fully clamped. For simplicity, and to preclude

35

the development of very large deformations, the plate material is assumed to behave elastically; however, large displacement effects are included in the computation. The impacting body is a cylinder 1 inch in radius and 4 inches long (L/D=2). The cylinder material is gelatin with 10% porosity, with density 0.0344 pounds per cubic inch. No mechanical properties are available from Reference [23]; for this calculation, we assume a shear modulus of 100. psi, linear and quadratic bulk moduli of 1000. psi, and a material strength of 10. psi.

The impact velocity for the event has been chosen to correspond to one of the rigid target cases reported in Reference [23], at 4724.4 inches per second (120 meters/second). The approximate "squash-up" time (the time for the cylinder to travel a distance equal to its initial length) is therefore 0.0008466 seconds.

For the finite element solution of this problem, 25 thin shell elements are used to represent the plate, and 48 eight-node hexahedra model the cylinder. Both of these models are coarse, since the objective is not to perform a detailed stress analysis of the event; the most important effects which the solution must capture are the transfer of momentum between cylinder and plate, and the tendency of the impacting body to spread over the surface of the target. In the structure mesh, a constant time step of 0.01 milliseconds (about 1.2% of the squash-up time) is used; for the soft body mesh, the time step is adjusted automatically and continuously within the program. It is important to notice that the resolution with which impact forces can be obtained from the solution is limited to the structure mesh time step, since the interface force calculations are performed only at the beginning of each time step.

Figure 3.1 gives an overview of the impact calculation, showing the boundaries of both bodies at times of 0.1 ms, 0.3 ms, 0.5 ms, and 0.7 ms. The configuration at 0.7 ms represents the final point of the solution, at which time the cylinder just begins to rebound from the surface of the plate. By this time,
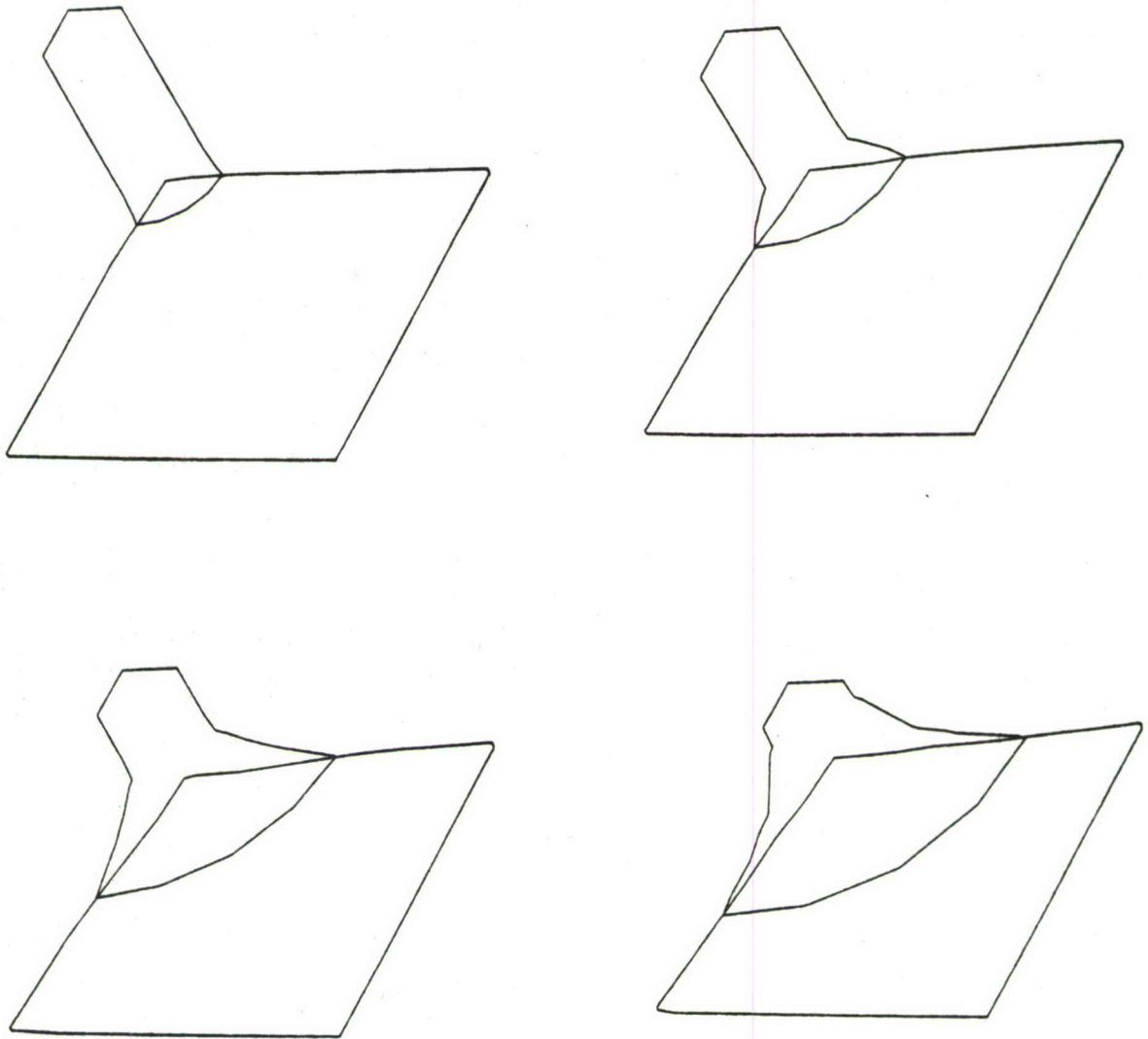
Figure 3.1.   Deformed Geometry of Plate and Cylinder at
0.1 ms, 0.3 ms, 0.5 ms, and 0.7 ms.

37

the plate has attained its maximum deflection and is vibrating upward. Obviously this same rebounding does not occur for the rigid plate tests reported in Reference [23].

Figure 3.2 shows the deformed geometry of the cylinder at intervals of 0.05 ms. The body is about 70% consumed during the solution, as shown in the side view of Figure 3.3 (just at the point of rebound). This is at least qualitatively correct: the deflection of the plate reaches about 0.75 inches at 0.65 ms, which means that the rear surface of the cylinder, traveling at its initial velocity, would reach the plate at approximately 1.0 ms. The soft-body mesh has been rezoned five times during the interval shown in the Figure. The usual cause for rezoning is exceedance of the time-step-change tolerance, which is set to 0.75; in two instances, the element volume change criterion has been activated due to extreme compression of elements adjacent to the impact surface.

Time histories of the interface forces, vertical momentum of the cylinder, and plate central deflection are given in Figures 3.4 through 3.6, respectively. The theoretical impact force during steady flow, for the quarter of the cylinder modeled, is [23]:

$$F = \frac{1}{4} \rho A u_o^2 = 1558.4 \text{ lb.}$$

This force level is indicated by a horizontal line in Figure 3.4. During the initial stages of the impact, the total vertical force oscillates about the theoretical value, and becomes larger at later times. The following important points should be noted:

o the total forces plotted in Figure 3.4 represent values at the beginning of structure time steps only, and therefore are limited in resolution (note that the momentum curve for the cylinder, Figure 3.5, is much smoother, since this output is generated several times per increment);
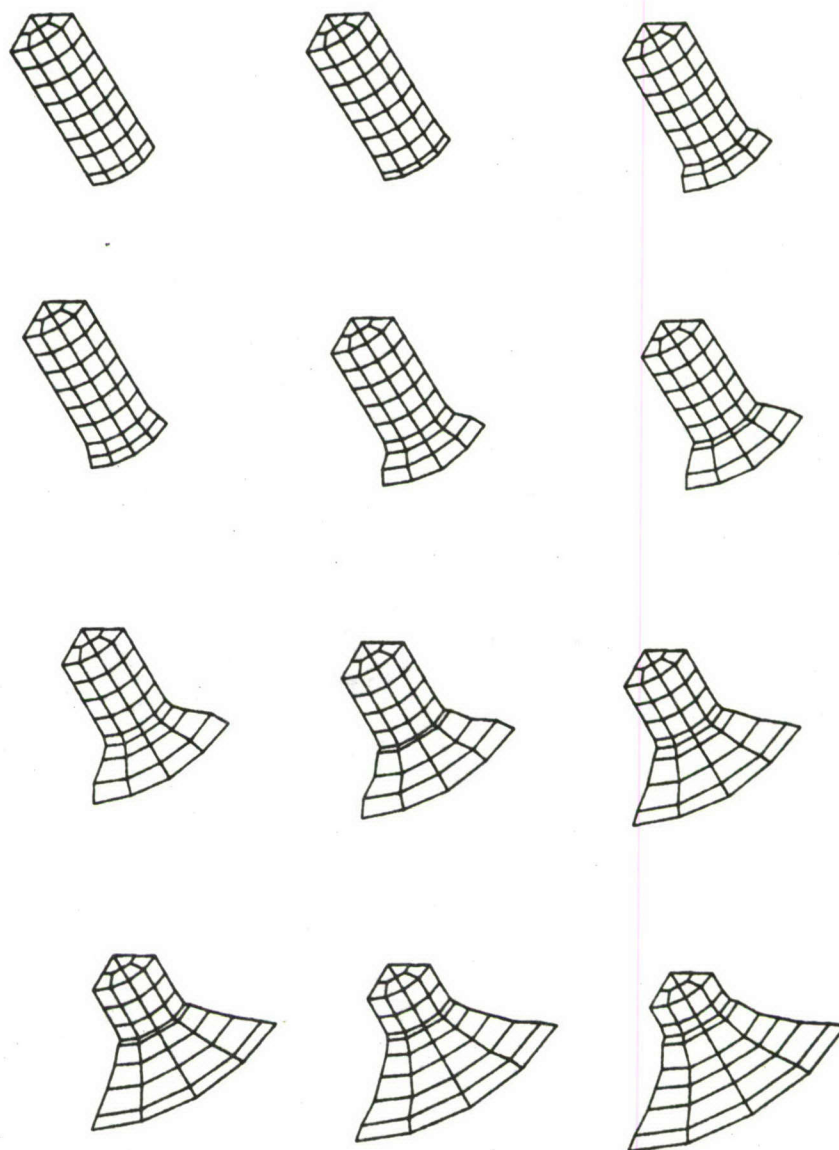
Figure 3.2. Soft-Body Mesh at Time Intervals of
0. 05 Milliseconds.

39
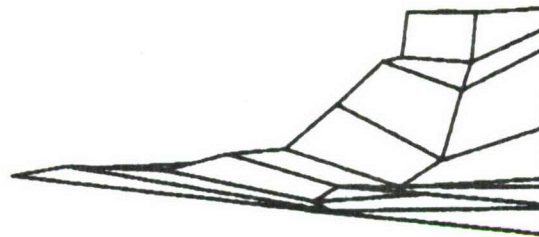
Figure 3.3.   Elevation View of Cylindrical Body at
              0.7 Milliseconds.

40
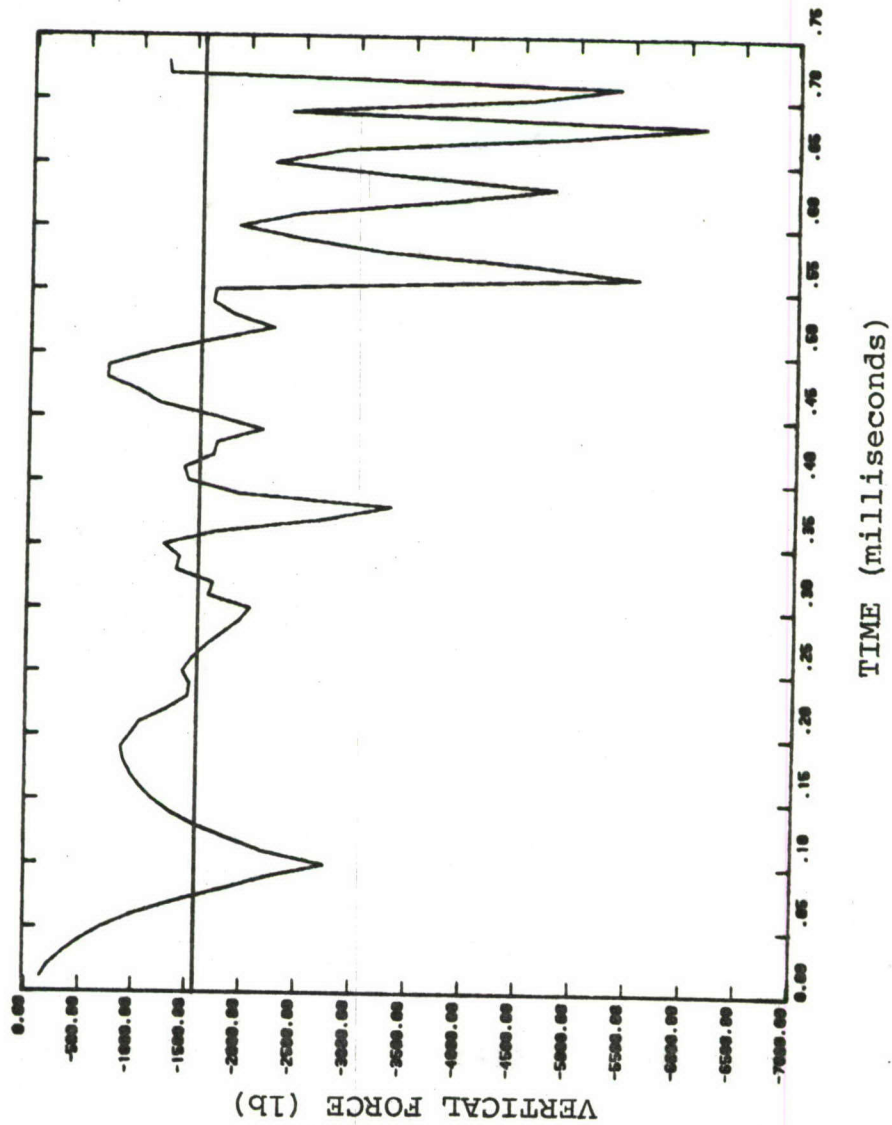
Figure 3.4.   Impact Force versus Time.

41
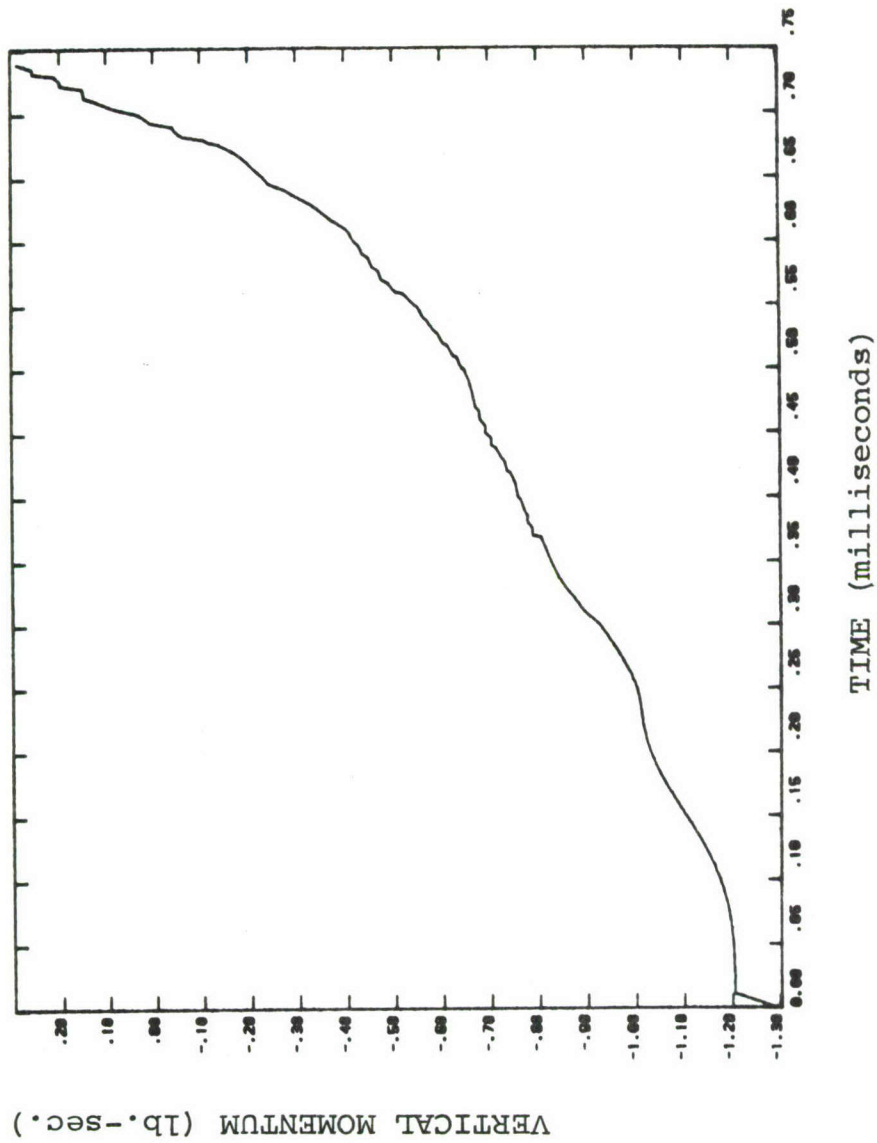
Figure 3.5. Vertical Momentum of Soft Body Mesh versus Time.

42
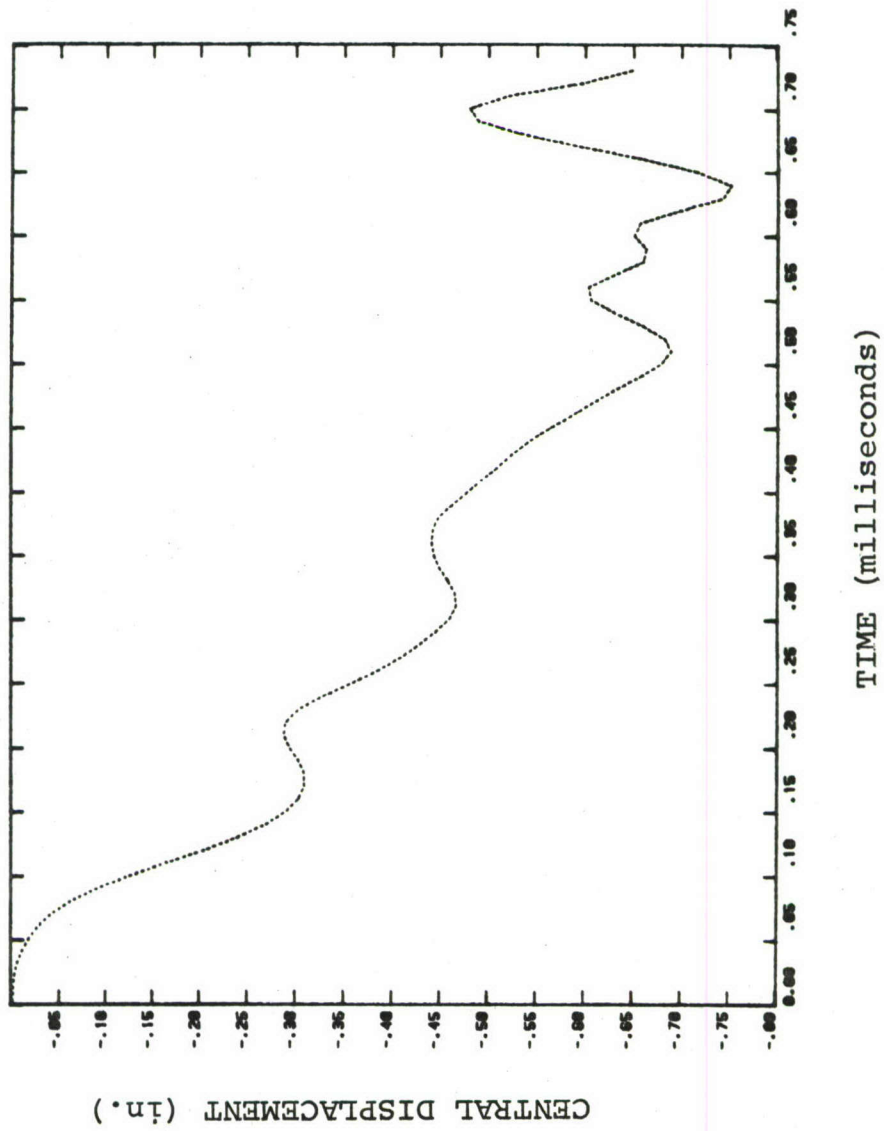
Figure 3.6. Plate Central Deflection versus Time.

43

o the shell elements used for the plate are bilinear and
    give a rather coarse displacement shape; as individual
    elements change their orientation abruptly from step to
    step, unrealistic oscillations occur in the vertical
    forces which would not be expected with higher-order
    structural finite elements;

o the increase in vertical forces late in the event is
    directly related to the displacement behavior of the
    plate; note, for example, that the peaks in the force
    curve correspond to times at which the plate velocity has
    changed sign, and the center of the plate moves upward.

As Figure 3.6 shows, the plate central deflection reaches a
maximum near 0.65 ms, and exhibits some higher-frequency
oscillations prior to that time. These oscillations are rather
mild during the early stages of the impact, but become more
pronounced and shorter in duration as the deflections increase,
presumably due to membrane stiffening effects.

The results of the normal impact case above are encouraging,
since the calculations are well-behaved and yield reasonably
accurate information about the impact interface forces. Further
testing needs to be performed to address the following questions:

(1) Does a finer (or higher-order) structural model lead to
improved impact force predictions?

(2) Does the solution predict accurately the distribution of
loading over the target? Useful experimental results exist for
aircraft transparencies, with loading areas recorded using high
speed photography, which can be used to verify this aspect of the
analysis technique.

(3) How accurately must the bird (or bird substitute) material be
characterized in order to obtain reasonable predictions of impact
pressures, spreading, and load-response coupling?

We feel that the most serious limitations which exist at
present have not to do with the analytical model, but with the

lack of reliable data for materials characterization. While mechanical property values may not be critical to the calculation of steady flow phase impact forces, inaccurate material modeling will affect the spatial distribution of loading on the target, particularly for more complex geometries. In many transparency impact situations, the interaction between structural response and the spatial distribution of the loading is felt to be a critical factor in determining the transparency dynamic response.

SECTION 4
COMPUTER CODE DESCRIPTION

This Section contains a brief description of the computer implementation of the soft-body impact analysis. Information contained in the section will be of interest to programmers or other persons involved in further development, conversions of the computer code to other machines, or interfacing of the analysis routines with other structural dynamics programs. Information about the routine usage of the computer program can be found in Section 5.

4.1 OVERVIEW

The computer code which implements the soft-body impact analysis consists of a set of input, analysis and output subroutines, which can be executed either as a stand-alone program (with a suitable main program inserted), or coupled with existing programs for structural dynamic analysis. For convenience, we refer to this collection of software as the SBI (Soft-Body Impact) code. The existing finite element program MAGNA [16] has been used as the structural analysis module, it is simply referred to by name in the discussion to follow.

The SBI code is written in ANSI FORTRAN 77, with the exception of one subprogram (CPUSEC) which must be provided for each target machine. The program has been compiled and executed on CDC, CRAY and VAX computers without incident. The machine dependencies in SBI are limited to three categories:

(1) the function subprogram CPUSEC, which returns the CPU time from the start of the job, is machine-specific;

(2) IMPLICIT DOUBLE PRECISION statements are included in each subprogram which uses variables of real data type; these IMPLICIT statements contain the leading characters "C-DBL", which can be removed to activate the statements on machines with short word lengths; and

(3) file OPEN statements used in a driver program in the stand-alone mode may require different file name parameters for the printed output file (e.g., OUTPUT on CDC, $OUT on CRAY, TT: or none on VAX-11).

The remaining subsections describe the major components of the SBI code, and the routines which are necessary to interface the hydrodynamics analysis with a structural analysis code.

4.2 MODULE AND COMMON BLOCK SUMMARIES

The hydrodynamics (SBI) code consists of five primary program modules: input, analysis, rezoning, output and restart. These modules can be invoked in virtually any order, except that the input module must be called once at the beginning of a new problem. Entry points into the first three modules are unique; that is, there is only one way of invoking them:

Input	– SUBROUTINE INCTRL
Analysis	– SUBROUTINE FLSOLV
Rezoning	– SUBROUTINE RZMAIN

The remaining two modules consist of utility subroutines, any of which may be called as needed:

Output	– SUBROUTINEs PRTXYZ, PRTVEL, PRZONE, MSHMPO
Restart	– SUBROUTINEs FLSAVE, FLREST

In the stand-alone mode of operation, a driver program is needed to control the execution of the input, analysis, output and restart modules. The rezoning operation is normally called as needed under control of the analysis segment. When the SBI analysis is coupled with a structural analysis program, the interface subroutines perform this control function. In the coupled mode, the structural analyzer acts as the control program, calling the SBI routines for input, output, analysis or restart functions as needed.

Communication between the SBI modules and a calling program is accomplished in two ways. First, a real array is supplied to the SBI routines as working storage, which is dynamically allocated as needed. Second, a series of ten COMMON blocks is used to retain key solution parameters. The contents of these blocks are described below, including the modules in which they are defined and used. Single-letter codes used to identify the modules are: (I)nput, (A)nalysis, (O)utput, (R)estart, re(Z)one. When variables in COMMON must be defined in the driver program or the structural analyzer interface, a 'D' code appears.

COMMON Block /PROBID/

Name	Defined	Used	Description
ITITLE	I	IAOR	Alphanumeric title (CHARACTER*80)

COMMON Block /CTLBLK/

Name	Defined	Used	Description
NWORK	D	IARZ	Length of working array (real words)
LWORK	A	IARZ	Length of working array (real words) available for use in solution phase
LWA	I	IA	Last word allocated during input
IFILL	–	–	Reserved space, 20 integer words
NUMNOD	IZ	IAORZ	Current number of nodes in mesh
NUMEL	IZ	IAORZ	Current number of elements in mesh
NUMMAT	I	IARZ	Number of materials defined
NVAR	I	IAORZ	Number of solution variables (normally 11 – 3 velocities, density, 6 stresses, and effective strain parameter)
NUMRWC	I	IAR	Number of rigid-wall boundary conditions
NUMCON	I	IAR	Number of contact (moving surface) boundary conditions
NREF	I	IZR	List of length (3) giving the number of rezoning reference stations in each coordinate direction.
NUMSBC	I	IAR	Number of symmetry boundary conditions
NFILL	–	–	Reserved space, 29 integer words
LCORD	IZ	IARZ	Location of coordinate data block in the main working array
LSVAR	IZ	IARZ	Location of solution variables data block
LCONN	IZ	IARZ	Location of element connection data block
LMATL	I	IAR	Location of properties data block
LRWBC	I	IAR	Location of rigid-wall b.c. data block

Name	Defined	Used	Description
LCNBC	I	IAR	Location of contact b.c. data block
LBODY	I	IAR	Location of body forces data block
LREFP	I	IAR	Location of rezoning reference position data block
LSYBC	I	IAR	Location of symmetry b.c. data block
LFILL	−	−	Reserved space, 29 integer words

COMMON Block /SOLPAR/

Name	Defined	Used	Description
TIME	IA	AOR	Current time value
TMAX	I	AR	Maximum time value
TREST	IR	R	Last time at which restart file created
TPOST	IO	OR	Last time at which results file created
DTMIN	I	AR	Lower limit on solution time step
DTMAX	I	AR	Upper limit on solution time step
FILL	−	−	Reserved space, 10 real words
INCR	IA	AR	Current time step number
INCMAX	I	AR	Upper limit on number of time steps
IREST	I	DR	Flag for restart output (0=off,1=on)
IPOST	I	DR	Flag for results output (0=off,1=on)
MFILL	−	−	Reserved space, 10 integer words

COMMON Block /TOLBLK/

Name	Defined	Used	Description
DTFRAC	I	AR	Fraction of critical time step to be used in the solution, typically 0.7-0.9
VOLTOL	I	AR	Relative volume change permitted before rezoning is invoked, typically 0.1-0.5
SDTTOL	I	AR	Relative change permitted in critical time step before rezoning is invoked, typically 0.25-0.5
HGDAMP	I	AR	Hourglass damping fraction. This is the fraction of critical damping used to suppress hourglassing instabilities, typically 0.05-0.5
DFACT	I	AR	Time step reduction due to hourglass damping. DFACT is applied to the stable time step to preserve stability. Set to SQRT(1-HGDAMP**2)-HGDAMP

COMMON Block /FLSPAR/

Name	Defined	Used	Description
DT	A	AR	Current time step
TQUIT	D	AR	Maximum time value used to force an exit from the analysis module at intermediate times in the solution. Overrides TMAX.
IFIRST	D	AR	Flag for first pass into analysis module; forces initial conditions to be applied. =1 for first pass, =0 afterward

COMMON Block /FILNAM/

Name	Defined	Used	Description
NIN	D	IR	Logical unit number for data input
NOUT	D	IAORZ	Logical unit number for printing

COMMON Block /FLFILE/

Name	Defined	Used	Description
IUPOST	D	OR	Logical unit number for results file
IUREST	D	R	Logical unit number for restart file
IUWORK	D	RZ	Logical unit number for scratch file to be used during rezoning. The file must be opened for unformatted sequential access prior to entering either the analysis module or the rezoning module.

COMMON Block /FSINTR/

Name	Defined	Used	Description
NSNODE	I	AR	Number of nodes in interface set
NSELEM	I	AR	Number of interface (contact) elements
NFSFIL	D	IAR	Direct access file for interface data
NIDREC	D	IAR	File record for interface node numbers
NCNREC	D	IAR	File record for interface element data
NNCREC	D	IAR	File record for interface geometry data
NNVREC	D	IAR	File record for interface velocity data
LSNID	I	AR	Location of interface array ISNID
LSCON	I	AR	Location of interface array ISCON
LXYZS	I	AR	Location of interface array XYZS
LVELS	I	AR	Location of interface array VELS
LTR	I	AR	Location of interface array TR
LXMIN	I	AR	Location of interface array XMIN
LXMAX	I	AR	Location of interface array XMAX
LYMIN	I	AR	Location of interface array YMIN
LYMAX	I	AR	Location of interface array YMAX
LZMAX	I	AR	Location of interface array ZMAX
LFSEND	I	AR	End of data area for interface arrays

COMMON Block /WRKSPC/

Name	Defined	Used	Description
WORK	A	A	Scratch space for fixed-dimension arrays used in element calculations. Block must be at least 136 words in length.

53

COMMON Block /PCOM1 /

Name	Defined	Used	Description
PCOM	A	AZ	Scratch space used to compress rezoning station data prior to beginning a rezone. Block must be at least 180 words in length.

COMMON Block /DP   /

Name	Defined	Used	Description
IDP	D	IARZ	Flag for DOUBLE PRECISION usage. =1 if variables of type REAL are of the same length as type INTEGER; =2 if type REAL variables are twice as long as INTEGERs.

Typically, parameters NWORK, IFIRST, NIN, NOUT, IUPOST, IUREST, IUWORK and IDP are defined in the driver program prior to calling any of the SBI segments. The intermediate time parameter TQUIT is defined prior to entering the analysis module each time it is called. For execution in the coupled mode, the following parameters in block /FSINTR/ must be defined prior to entering the input module: NFSFIL, NIDREC, NCNREC, NNCREC, and NNVREC.

Entry to the SBI modules requires the following subroutine calls (the main working array is called "A"):

1. Input Module

```
----------------
    CALL INCTRL ( A, MODE )


    where MODE = 0 for stand-alone execution, 1 for coupled mode
```

2. Analysis Module

```
------------------
    CALL FLSOLV ( A, IFLAG, IRZOPT, MODE, CDATA )


    where IFLAG is an output status flag (0 for normal completion,
               1 for error(s) in element geometry, 2 for errors
               during rezoning, 3 for insufficient storage)
         IRZOPT > 0 forces rezoning upon exit, = 0 specifies
               rezoning only as necessary
         MODE = 0 for stand-alone execution, 1 for coupled mode
         CDATA is an array containing interface data for the
               coupled mode (see subsection 4.8)
```

3. Output Module

```
----------------
    CALL PRTVEL ( NUMNOD, NVAR, A(LSVAR) )
    CALL MSHMPO ( NUMNOD, A(LCORD), NUMEL, A(LCONN), NVAR,
               A(LSVAR), NREF, A(LREFP), IUPOST, INCR, TIME )
```

4. Restart Module

```
----------------
    CALL FLSAVE ( A, IUREST )
    CALL FLREST ( A, IUREST )
```

5. Rezoning Module

```
------------------
    CALL RZMAIN ( NUMNOD, NUMEL , NVAR , NREF , A(LREFP),
               NWORK , A       , LCORD, LSVAR, LCONN   ,
               IERROR, IUWORK, NOUT )


    where IERROR is an output status flag (see subsection 4.6)
```

4.3 INPUT MODULE

The input module of the SBI code controls the reading of problem data describing the hydrodynamic finite element mesh (see Section 5.3). During the input procedure, the input module performs memory allocation for data as it is read, and prestores addresses at which the mesh interface data will be stored in the coupled execution mode.

Entry to the input module is achieved by the single subroutine call

        CALL INCTRL (A, MODE)

in which A is the main working array, and MODE is a flag indicating the execution mode (MODE=0 for stand-alone, >0 for coupled mode). The call to INCTRL causes the following input data blocks to be read from file NIN:

     1. TITL - problem title
     2. PARA - solution parameters
     3. TOLE - tolerances for rezoning and hourglass forces
     4. PROP - physical properties
     5. REFE - rezoning reference positions*
     6. SYMM - symmetry boundary conditions
     7. RIGI - rigid wall boundary conditions
     8. CONT - contact (structure interface) data, if MODE>0
     9. BODY - prescribed body forces
    10. NODE - nodal coordinate data
    11. INIT - initial conditions
    12. ELEM - finite element definitions

Parameters, tolerances, sizing data and address information are stored in COMMON blocks (see subsection 4.2) by the input module. Remaining data are stored sequentially in the main working array "A", as summarized below.

---

* REFE block input is particular to rezoning algorithm I (see subsection 2.6.2), but is required when either algorithm is used.

Location	Data Type	Description
LMATL	Real	Physical properties data
LREFP	Real	Rezoning reference stations
LSYBC	Real	Symmetry boundary condition data
LRWBC	Real	Rigid-wall boundary condition data
LBODY	Real	Body forces
LCORD	Real	Nodal coordinates
LSVAR	Real	Solution variables, with initial conditions applied
LCONN	Integer	Element connections

Data are read and stored in the above sequence, regardless of the order of data blocks in the input file. Note that data blocks which remain unchanged when rezoning is performed are stored first. The coordinate, solution variable and element connection blocks may change in size when a rezone occurs.

In the coupled mode, the input module also processes interface definition data (CONT data block); this data is organized in its final form and written directly to random access storage for use by the interface subroutines. File and record numbers are defined in the calling program and are stored in COMMON block /FSINTR/.

4.4 RESTART MODULE

The SBI restart module is used to save or restore problem data at an intermediate stage in the problem solution. Two subroutines exist for this purpose:

- FLSAVE (save data for restart), and
- FLREST (restart from stored data).

Entry to these routines requires the following subroutine calls:

- CALL FLSAVE (A, LUNIT)
- CALL FLREST (A, LUNIT)

Here A is the main working array and LUNIT is the number of the sequential file to which the restart information is written.

The restart file consists of two records, containing the contents of the major common blocks for the SBI code as well as the main working array A. The two restart records are written as follows:

```
WRITE (LUNIT) NWORK , LWORK , LWA    , IFILL ,
              NUMNOD, NUMEL , NUMMAT, NVAR   , NUMRWC,
              NUMCON, NREF   , NUMSBC, NFILL ,
              LCORD , LSVAR , LCONN , LMATL , LRWBC ,
              LCNBC , LBODY , LREFP , LSYBC , LFILL ,
              TIME   , TMAX   , TREST , TPOST , DTMIN ,
              DTMAX , FILL   ,
              INCR   , INCMAX, IREST , IPOST , MFILL ,
              DTFRAC, VOLTOL, SDTTOL, HGDAMP, DFACT ,
              DT     , TQUIT , IFIRST
LASTWD = LCONN + 8*NUMEL/IDP - 1
WRITE (LUNIT) (A(I),I=1,LASTWD)
```

Interface data for operation in the coupled mode must be saved elsewhere for restart, since the data required for linking the SBI solution with a particular structural analyzer will be highly program-dependent.

4.5   SOLUTION MODULE

The SBI solution module performs a segment of the explicit, hydrodynamic solution, consisting of a specified time interval or a predetermined number of time steps (whichever comes first). The organization of input and output data is as described in subsections 4.3 and 4.7. However, since the solution package can invoke the rezoning procedure, the input and output array sizes may differ.

The solution module is entered through the subroutine call:

CALL FLSOLV (A, IFLAG, IRZOPT, MODE, CDATA)

Formal parameters for the module are as follows:

A   = Main working array, described in subsection 4.3.

IFLAG   = Output status flag
          =0, Normal completion (TQUIT or INCMAX reached)
          =1, Error(s) in element geometry detected
          =2, Error(s) occurred during rezoning operation
          =3, Working storage allocation exceeded

IRZOPT = Switch for rezoning of final solution
          =0, Rezoning is performed only as necessary
          >0, Forces rezoning before leaving solution module

MODE   = Switch for execution mode
          =0, Stand-alone
          =1, Coupled with structural analysis

CDATA = Structural interface data, used only when MODE=1

The actual problem data are supplied to the solution routines in the work array A, and in the COMMON blocks described in

59

subsection 4.2. The contents of array CDATA, which describe the structure model interface, are summarized in subsection 4.8.

A hydrodynamic solution may be performed either through a single call to FLSOLV, or by a series of calls; minor differences may occur between these two methods when rezoning is performed (particularly if IRZOPT>0). The usual reasons for performing a series of calls to the solution module are to allow results and restart data to be stored at intervals throughout the analysis, or to force results to be generated at regular intervals during the solution.

4.6 REZONING MODULE

The SBI rezoning module performs a complete reorganization of the finite element grid for the hydrodynamic solution. This module is suitable for use with other forms of finite element data as well, provided it conforms to the input conventions described below. In effect, the input to the rezoner is a finite element model composed of eight-node hexahedral elements, together with certain information to control the rezoning process; the output is a different finite element model with minimally-distorted elements.

There are two primary limitations which exist in the rezone module. First, the model must be composed entirely of eight-node brick elements. Second, the number of nodal solution variables is limited to 11 by fixed DIMENSION statements in several of the rezoning subroutines.

Entry to the rezoning package using rezoning algorithm I (subsection 2.6.2) requires a single subroutine call having the form:

```
CALL RZMAIN ( NUMNOD, NUMEL , NVAR  , NREF  , XREF  ,
              NWORK , A      , LCORD , LSVAR , LCONN ,
              IERROR, LUNIT , NOUT                    )
```

Formal parameters for the routine are:

NUMNOD (input)= Number of nodes in input mesh
        (output)= Number of nodes in output mesh

NUMEL   (input)= Number of elements in input mesh
       (output)= Number of elements in output mesh

NVAR   (input)= Number of solution variables per node,
          $0 < NVAR < 12$

NREF   (input)= Integer array of length 3 containing the number of reference positions in the $X1$, $X2$, and $X3$ directions, respectively

XREF   (input)= Real array containing the reference station positions in ascending order along each of the coordinate directions. $XREF(i,j)$ is

61

the j-th reference station in the i-th direction. The leading dimension of XREF must be MAX(NREF(1),NREF(2),NREF(3))

NWORK (input)= Number of floating-point words available in the working array 'A' (see below)

A   (input)= Primary data array, containing the nodal coordinates, solution variables, and element connections, at the starting locations described below (see LCORD, LSVAR, and LCONN). These arrays must be stored in the order stated above.

       (output)= Primary data array, containing the same data for the new finite element mesh.

LCORD (input)= Starting location for nodal coordinate data in array 'A' for the input mesh

       (output)= Starting location for nodal coordinate data in array 'A' for the output mesh

LSVAR (input)= Starting location for solution variables data in array 'A' for the input mesh

       (output)= Starting location for solution variables data in array 'A' for the output mesh

LCONN (input)= Starting location for element connection data in array 'A' for the input mesh

       (output)= Starting location for element connection data in array 'A' for the output mesh

IERROR (output)= Exit error code, defined as follows:

= 0   , Normal completion

= 1   , Number of nodes illegal

= 2   , Number of elements illegal

= 3   , Number of solution variables illegal

= 4   , Number of reference stations illegal for one or more coordinate directions

= 5   , Reference station values for one or more directions not in ascending order

= 10   , Input data (LCORD, LSVAR, LCONN) out of sequence

= 20   , Parameters LCORD, LSVAR and/or LCONN
         inconsistent with number of nodes,
         elements or solution variables
         declared

= 100 , Storage exceeded before rezoning

= 1000, Storage exceeded during generation of
         new node points

= 1010, Storage exceeded during generation of
         new elements

= 2000, Error(s) detected during element fill
         operation

LUNIT   (input)= Logical unit number of scratch file for use
                  in the rezoner. This file must be opened
                  for unformatted, sequential I/O prior to
                  calling RZMAIN, and may be discarded or
                  overwritten afterward.

NOUT   (input)= Logical unit number for line printer
                  output, including error messages and
                  rezoning statistics. If NOUT=0, no printed
                  output is generated by the rezoning module.

No information is passed to the rezoning package through COMMON,
to facilitate its use as a utility module in other applications.

Using rezoning algorithm II, entry to the rezoning module
requires the following subroutine call:

```
CALL RZMAIN (NUMNOD, NUMEL , NPASS, LENGTH, IERROR,
             NVAIR , XYZ   , VAR  , NCON  , WORK  ,
             IWORK1, IWORK2, LUNIT)
```

Formal parameters for the routine are:

NUMNOD   (input)   = Number of nodes in mesh

NUMEL   (input)   = Number of elements in mesh

NPASS   (input)   = Number of rezoning passes to be performed
                     (normally 1)

LENGTH   (input)   = Length of work array IWORK2 (see below

IERROR   (output) = Exit error code, defined as follows:
                    =0, normal completion
                    =1, error(s) in remapping solution
                        variables
                    = -1, length of IWORK2 must be increased by
                        n words to perform rezoning

NVAR   (input) = Number of solution variables (normally 11)
XYZ   (input) = Cartesian coordinates of nodes (3xNUMNOD)
         (output) = Coordinates in rezoned mesh
VAR   (input) = Solution variables array (NVAR x NUMNOD)
         (output) = Remapped solution variables
NCON   (input) = Element connectivity (8 x NUMEL)
WORK   (input) = Work array at least 3 x NUMNOD words in
                    length
IWORK1   (input) = Work array at least NUMNOD integer words
                    in length
IWORK2   (input) = Work array of estimated length at least 8
                    x NUMEL integer words
LUNIT   (input) = Logical unit number of scratch file
                    available for use by RZMAIN. This file
                    must be opened for unformatted, sequential
                    I/O prior to calling RZMAIN.

4.7 OUTPUT MODULE

The SBI output module consists of four subroutines which may be called at any time to produce printed or file output. Three of these (PRTXYZ, PRTVEL, MSHMPO) are used routinely for output of analysis results; the fourth, PRZONE, is a diagnostic routine which summarizes intermediate results during rezoning.

PRTXYZ prints a listing of current nodal coordinates on a designated output file. Since the hydrodynamic mesh is updated at each time step, this output describes the deformed geometry of the finite element model at the current time. PRTXYZ is called as follows:

    CALL PRTXYZ (NUMNOD, XYZ, NOUT)

in which NUMNOD is the number of nodes currently defined for the mesh, XYZ is the coordinate data in a 3 by NUMNOD array, and NOUT is the logical unit number for output.

PRTVEL is similar to PRTXYZ, but prints velocity information in a similar format. The only important differences are in the header information printed by the routines, and in the storage format expected. Because the velocities are normally stored together with other solution variables, the calling sequence for PRTVEL includes NVAR, the number of solution variables; the first three entries of each row of the NVAR by NUMNOD array VAR. The call to PRTVEL looks like:

    CALL PRTVEL (NUMNOD, NVAR, VAR)

Output is written to the unit number NOUT as defined in COMMON block /FILNAM/.

MSHMPO generates postprocessing file output in a form similar to the MAGNA results file "MPOST" [16]. The calling sequence for MSHMPO is:

    CALL MSHMPO (NUMNOD, XYZ, NUMEL, NCON , NVAR, VAR,
                  NREFX , XR , NOUT , INCRNO, TIME   )

Parameters NUMNOD, XYZ, NVAR, VAR and NOUT are as described above

65

for the other output routines. NUMEL defines the number of elements, and NCON(8,NUMEL) contains the element connections. NREFX(3) contains the number of rezoning reference stations in each coordinate direction, and XR the actual reference station coordinates. The minimum and maximum reference position values are written as eight extra nodes with the postprocessing file output, so that plotting routines which examine the coordinates for minimum and maximum coordinate values to determine scaling of plots will produce sequences of plots having the same scaling characteristics. INCRNO and TIME define a time step number and time value to be included in the output headers on the results file.

Subroutine PRZONE is normally used only for debugging. It must be called from subroutine REZONE in the rezoning module, by modifying the source code in that module.

4.8 MESH INTERFACE CODE

The mesh interface utilities consist of several subroutines which link the SBI code to a nonlinear structural analysis program, for the purpose of performing hydrodynamic impact analysis. These routines fall into three categories:

(a) Interface data input;

(b) Interface geometry construction; and

(c) Analysis interface.

Each group of utility routines are described briefly below. The descriptions are intended only as a general guide to interfacing the SBI code with a structural analysis program, since the details of doing so are almost completely program-dependent.

The interface data input routines include FSDATA, FSINPT, FSICON, and FSNIDS. FSDATA acts as a driver for the SBI input module, opening the necessary files and defining necessary parameters in COMMON (see subsection 4.2). The remaining routines are called from the input module (subsection 4.3) whenever MODE=1, to read the structural interface data. The data defined in these input routines consists of interface element definitions in terms of the nodes of the structural model (see subsections 2.7 and 5.3 for details). The structural analysis program typically calls FSDATA once during the input phase of execution.

Interface geometry construction is performed in subroutines FSPUTC, FSGETC, and FSGETU. FSPUTC is called by the structures program to save the nodal coordinates on file as soon as they are available. FSGETC must be called later, after the interface data have been read. The result of this stage, which is performed only once, is to save original coordinates for the interface elements. These will be used repeatedly in the analysis phase to compute current positions of the interface elements. This phase is unnecessary if a relative (updated Lagrangian) description of

motion is used in the structural analysis, since updated nodal coordinates should be available at any time during the solution.

The analysis interface consists of several subroutines (FLCALC, FSXYZV, FSTRAN, FSCONT, and FSFORC) which are called at each major (structure) time step of the coupled solution. The primary entry point is FLCALC, which controls the assembly of structure mesh data for the impact analysis, calls the solution module, and controls the retrieval of interface nodal forces. The secondary routines are described briefly below.

> FSXYZV - retrieves coordinate and velocity information for structure nodes which lie on the interface.

> FSTRAN - performs preliminary geometric calculations for all interface elements based on their current geometry; these include transformations to local coordinates, and storing of minimum and maximum coordinate values.

> FSCONT - this routine is called from FLSOLV (solution module) whenever MODE>0, to perform the actual interface contact calculations. The results are coordinates and velocities updated to reflect the existing contact conditions, and nodal values of the interface forces.

> FSFORC - accepts the interface forces computed in FSCONT, and generates nodal forces to be applied to the structural mesh during the current time step.

The analysis interface routines FSDATA and FLCALC also use the restart module subroutines to save and restore hydrodynamic mesh data as needed during the solution.

Interface data which is passed to the solution module during a coupled analysis is arranged sequentially in the array CDATA (see subsection 4.5). This data is assembled by FLCALC (above), and contains the following lists and arrays:

ISNID - Structure model nodes by sequential order as they
         appear on the interface

ISCON - Interface panel connections (4 per element)

XYZS   - Interface nodal coordinates

VELS   - Interface nodal velocities

TR   - Global-to-local transformation for each interface
         element

XMIN   - Minimum local 'X' coordinate for each interface
         element

XMAX   - Maximum local 'X' coordinate for each interface
         element

YMIN, YMAX, ZMAX - (Similar to XMIN and XMAX)

HFORCE - Area for output of interface nodal forces

All of the above arrays except HFORCE are input to the solution
module; there size is solely dependent upon the number of nodes
and elements on the structural/hydrodynamic interface, and
therefore does not change over the course of the solution. The
interface element data is prestored before the contact analysis
to save time recomputing these parameters, at the expense of some
array storage space. The total array storage necessary is
10x(number of nodes on interface) + 18x(number of interface
elements).

SECTION 5
PROGRAM OPERATION

This section provides a brief description of procedures for operating the soft body impact analysis code. Both stand-alone and coupled modes of execution are discussed; the coupled mode is described with reference to a particular structural dynamics program, MAGNA, which has been linked with the hydrodynamic analysis during the present project. Subsection 5.3 summarizes the input blocks and parameters necessary for execution in either mode.

5.1 OPERATION IN STAND-ALONE MODE

For stand-alone execution of the hydrodynamic analysis, it is necessary only to provide the proper input files to the code and to initiate execution. Restart and postprocessing files must be saved on some computers after execution is completed. Some examples of job control language for CDC, VAX and CRAY computer systems are listed below.

Example 1. CDC-6600 or CYBER-175 under NOS (SUBMIT file)

```
/JOB
jobname.
/USER
CHARGE,*.
GET,INDAT=filename.
GET,SBILGO/UN=D820139.
DEFINE,MPOST.
SETTL,nnnnn.
SBILGO.
/EOR
/EOF
```

Example 2. VAX-11/780 under VAX/VMS (DCL command file)

```
$ SET VERIFY
$ ASSIGN/USER_MODE input_file_spec FOR005
$ ASSIGN/USER_MODE output_file_spec FOR006
$ ASSIGN/USER_MODE post_file_spec MPOST
$ RUN SBI
$ LOGOUT
```

Example 3.   CRAY-1/S or CRAY-X/MP under COS

```
JOB,JN=jobname,Tnnnn.
ACCOUNT,AC=acctno,US=userno,UPW=password.
ACCESS,DN=INDAT,PDN=filename1.
ASSIGN,A=FT95,DN=MPOST.
ACCESS,DN=SBI,PDN=SBI.
SBI.
SAVE,DN=MPOST,PDN=filename2.
EOF
```

Example 4.   CRAY-1/S under COS (United Information Services)

```
JOB,Tnnnn.
ACCOUNT,userno,password.
ASSIGN,A=FT95,DN=MPOST.
GET,INDAT=filename1/CI=TTY.
GET,SBI.
SBI.
PUT,MPOST/CO=BAT/D.
DFD,SBIDAY,R.
PUT,$OUT=SBIOUT/CO=BAT.
EOF
```

In the above job control files, the file INDAT contains problem input data in the form described in subsection 5.3. The results file is called MPOST in all of the examples. Since our primary interest is in the coupled mode execution, no standard procedures for accessing restart files have been developed. The driver for the hydrodynamics program can be modified to include the restart function; for each example above, appropriate ASSIGN, GET, SAVE or other file control statements must be added to define and save the restart file.

5.2   OPERATION IN COUPLED MODE

When the soft body impact analysis is coupled with a finite
element structural model, three additional steps are necessary:

- (1) provide the input file for the hydrodynamic mesh;
- (2) activate the appropriate option in the structures
program input; and
- (3) add the necessary file control statements to the job
control file.

In the case of coupled execution with MAGNA, the input file
for the soft body mesh (subsection 5.3) must be provided as the
formatted file "FSDATA". The input data must contain the CONT
input block, which defines the structural/hydrodynamic mesh
interface.

The option flag in MAGNA which activates the coupled impact
analysis is IOPT(18). IOPT option flags are discussed in Section
8.2 of the MAGNA user's manual. Necessary values of this and
other IOPT parameters are listed below.

IOPT	Columns	Value	Description
2	5 - 8	2	Selects transient dynamic solution
3,4	9 -16	2	Selects nonlinear analysis
18	68 -72	1	Selects coupled impact analysis

All other MAGNA options and parameters may be switched on or off
as needed to define the particular problem at hand.

The only additional job control statement needed for the
coupled analysis is a single statement to access the data file
FSDATA prior to execution; for example:

GET,FSDATA=filename.   (CDC under NOS)

$ ASSIGN/USER_MODE file_spec FSDATA   (VAX under VMS)

ACCESS,DN=FSDATA,PDN=filename.

73

```
          ASSIGN,A=FT40,DN=FSDATA.                    (CRAY-1 under COS)


          GET,FSDATA=filename/CI=TTY.
          ASSIGN,A=FT40,DN=FSDATA.                    (CRAY-1 at U.I.S.)
```

The restart and postprocessing files generated in the soft body
analysis are merged automatically with the corresponding MAGNA
files (NOREST, NRSTAP, and MPOST).

In coupled execution mode, additional data are also output
to the MPOST (postprocessor) file to provide a less complicated
summary of the hydrodynamic solution and to save the computed
values of interface forces. These data are interspersed with the
data blocks which usually appear on the MPOST file, and have the
following identifying keywords:

- CMOM :   center-of-mass and momentum data for the
             hydrodynamic mesh;
- HYDF :   hydrodynamic impact forces by structure node
             number; and
- TOTF :   total hydrodynamic forces in all directions.

The CMOM data is useful for visualizing the gross motion of the
soft body mesh, and for evaluating the momentum transfer to the
target. The HYDF data blocks provide sufficient information to
reconstruct the force-versus-time history generated in a fully
coupled solution, so that later simulations with minor parameter
changes can be run in an uncoupled mode using specified force
data. The force resultants from the TOTF blocks are useful when
compared with momentum data for the hydrodynamic model, since
the combination of the two provides a means of verifying that the
overall motion of the soft body is correct.

Output data and formats for the additional data blocks
mentioned above are as follows:

Block	Line(s)	Data Description	Format
CMOM	1	Block header	A4,1X
		SBI Increment no.	I5
		Time	E15.7
		X position of C.M.	E15.7
		Y position of C.M.	E15.7
		Z position of C.M.	E15.7
		X direction momentum	E15.7
		Y direction momentum	E15.7
		Z direction momentum	E15.7

Block	Line(s)	Data Description	Format
HYDF	1	Block header	A4,1X
		Structure incr. no.	I5
		Time	E15.7
		Structure node no.	I5
		X direction force	E15.7
		Y direction force	E15.7
		Z direction force	E15.7
HYDF	2,3,...	Blank	25X
		Structure node no.	I5
		X direction force	E15.7
		Y direction force	E15.7
		Z direction force	E15.7

Block	Line(s)	Data Description	Format
TOTF	1	Block header	A4,1X
		Structure incr. no.	I5
		Time	E15.7
		Blank	5X
		X force (total)	E15.7
		Y force (total)	E15.7
		Z force (total)	E15.7

The MPOST data blocks described in this Section can be extracted from the complete postprocessing file simply by searching for the appropriate block headers. It should be noted that the HYDF data block varies in length, depending on the number of structure nodes which experience non-zero impact forces. The end of the HYDF block is signalled only by the beginning of the TOTF block, which consists of a single line. If no impact forces exist for a particular time increment, only the TOTF line will appear in the file.

5.3 INPUT DATA DESCRIPTIONS

Problem input for the hydrodynamic analysis is supplied on a single formatted, sequential file. Data are separated into input "blocks"; each input block begins with a four (or more) character identifier, followed by the appropriate data; most end with a single blank line. Input blocks may appear on the file in any order. Comment lines may be inserted in the input file after the terminator (blank line) for any block, or at the beginning of the file. Comment lines have no special format, but should begin with a character other than A-Z to avoid duplicating valid input block names.

Data blocks recognized by the input module are as follows:

TITL - Problem title
NODE - Nodal coordinate data
ELEM - Finite element connection data
PROP - Physical properties data
BODY - Body forces
INIT - Initial conditions
SYMM - Symmetry or antisymmetry boundary conditions
RIGI - Rigid wall boundary conditions
PARA - Solution parameters and options
TOLE - Tolerance parameters
REFE - Reference position data for rezoning
CONT - Contact interface data (coupled execution only).

The content and format of each of these input blocks are described in the remainder of this subsection.

In general, the names of input data items follow FORTRAN typing conventions; that is, names beginning with letters I-N are of integer data type, and names beginning with A-H or O-Z are real (floating-point) data. Where character (alphanumeric) input is required (as in the block headers and title information), this is noted in the data description. Integer data may be entered anywhere within the fields provided, but may not contain decimal points or exponents (E). Floating-point data may be placed

anywhere within the data field, and may be expressed in virtually any format. For example, 20, 20.0, 2E1, an .2E2 all represent a numeric value of "20".

```
------------------------------------
B O D Y   F O R C E S   Input Block
------------------------------------
```

Header : BODY
End   : (1 Record)

Record Format #1 :

```
. . . .10 . . . .20 . . . .30
     XBODY      YBODY      ZBODY
```


Input Variables :

XBODY = X direction body force
YBODY = Y direction body force
ZBODY = Z direction body force

Notes :

  (1) - Body forces are defined in global Cartesian coordinate
        directions, and are constant throughout the solution.

  (2) - Body force data is expressed in terms of force per unit
        mass. For example, gravity acting in the negative "Y"
        direction is specified by setting YBODY = -g, where g
        is the acceleration of gravity in units consistent with
        the other problem data.

C O N T A C T   I N T E R F A C E   Input Block

Header : CONT
End   : blank line
Record Format #1 :

. . . . 5
   NSELEM

Record Format #2 : (repeat as needed to define NSELEM interface
                    elements)

. . 5 . .10 . .15 . .20 . .25 . .30 . .35
   IEL   IEG   ING   N(1) N(2) N(3) N(4)

Input Variables :

NSELEM = Total number of interface elements to be defined
IEL   = Interface element number (1,2,...,NSELEM)
IEG   = Element number increment for use in generation
ING   = Node number increment for use in generation
N(i)   = i-th connected node for interface element 'IEL'; note
         that node numbers refer to the nodes of the structural
         model

Notes :

  (1) - This input block defines that portion of the surface of
        the structural finite element model with which the soft
        body (hydrodynamic) model is expected to come into
        contact.

(2) - Interface elements normally correspond to exterior faces
of solid or shell elements in the structures model.

(3) - Node numbers which define the interface elements are
specified in terms of the node numbers in the structure
finite element model, not the hydrodynamic model.

(4) - Each interface element is defined by four corner nodes,
which appear in counter-clockwise order when viewed
from the outside of the contact surface.

```
------------------------------
E L E M E N T S   Input Block
------------------------------


Header : ELEM
End    : Blank line


Record Format #1 :


.  . 5
NUMEL


Record Format #2 : (repeat as needed to define NUMEL elements)


.  . 5 . .10 . .15 . .20 . .25 . .30 . .35 . .40 . .45 . .50 . .55
   IEL    IEG    ING   N(1)  N(2)  N(3)  N(4)  N(5)  N(6)  N(7)  N(8)


Input Variables  :


NUMEL   = Total number of elements to be defined
IEL     = Element number (1,2,...,NUMEL)
IEG     = Element number increment for use in generation
ING     = Node number increment for use in generation
N(i)    = i-th connected node for element 'IEL'
```

Notes :

(1) - The relative positions of local nodes 1-8 of an element
are indicated in the diagram below.
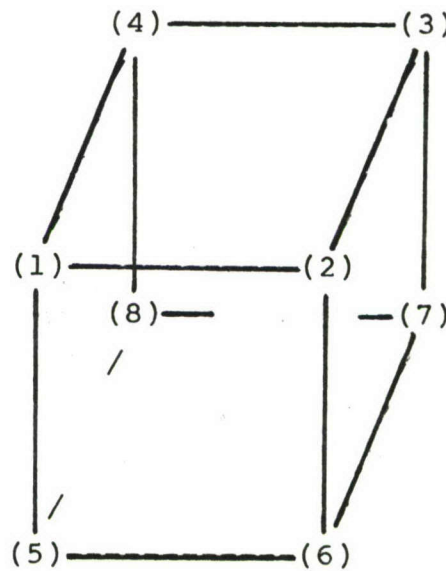
```
----------------------
N O D E S    Input Block
----------------------
```

Header : NODE
End   : Blank line

Record Format #1 :

```
. . . 5
  NUMNOD
```

Record Format #2 : (repeat as needed to define NUMNOD nodes)

```
. . . 5 . . .10 . . . . . . .20 . . . . . . .30 . . . . . .40
  NODE    INGEN        XCORD        YCORD        ZCORD
```

Input Variables :

NUMNOD = Total number of nodes to be defined
NODE   = Node number
INGEN = Node number increment for use in generation
XCORD = X coordinate
YCORD = Y coordinate
ZCORD = Z coordinate

Notes :

   (1) - Valid node numbers are integers from 1 through NUMNOD.

   (2) - Not all nodes between 1 and NUMNOD need be defined;
         however, storage is allocated for NUMNOD nodes, and the
         omission of large numbers of nodes may lead to excessive
         storage requirements.

   (3) - Nodal data may be read or generated in any order.

```
------------------------------------------------------------
I N I T I A L   C O N D I T I O N S   Input Block
------------------------------------------------------------
```

Header : INIT

End   : (1 Record)

Record Format #1 :


```
. . . .10 . . . . .20 . . . . .30 . . . . .40
     XVEL        YVEL        ZVEL        DENS
```


Input Variables :

XVEL   = Initial velocity in X direction
YVEL   = Initial velocity in Y direction
ZVEL   = Initial velocity in Z direction
DENS   = Initial value of material density

Notes :

   (1) - Initial conditions specified in this block apply to all
         nodes of the model.

   (2) - Except in special circumstances, the material density
         appearing above should agree with the reference density
         defined in the PROPerty input block.

```
-----------------------------------
P A R A M E T E R S    Input Block
-----------------------------------
```

Header : PARA
End   : (2 Records)

Record Format #1 :

```
. . . .10 . . . .20 . . . .30 . . . .40 . . . .50 . . . .60
     TIME       TMAX      TREST      TPOST     DTMIN     DTMAX
```

Record Format #2 :

```
. . . 5 . . .10 . . .15 . . .20
    INCR   INCMAX    IREST    IPOST
```

Input Variables :

TIME   = Time at start of solution
TMAX   = Maximum time value
TREST   = Time interval at which restart files are to be
          written (if IREST > 0)
TPOST   = Time interval at which results files are to be
          written (if IPOST > 0)
DTMIN   = Minimum time step value
DTMAX   = Maximum time step value
INCR   = Increment number at start of solution (usually 0)
INCMAX = Maximum number of time increments to be performed
IREST   = Restart save flag (>0 if restart files are to be
          created)
IPOST   = Results save flag (>0 if results files are to be
          created for plotting)

Notes :

  (1) - Presently, only the parameters TIME, TMAX, INCR, and
        INCMAX are used when a coupled analysis is performed.

```
------------------------------------
T O L E R A N C E S   Input Block
------------------------------------
```

Header : TOLE
End   : (1 Record)

Record Format #1 :


```
. . . .10 . . . .20 . . . .30 . . . .40
   DTFRAC    VOLTOL    SDTTOL    HGDAMP
```


Input Variables :

DTFRAC = Fraction of the maximum stable time step to be
          used in the solution (Default = 0.75)
VOLTOL = Relative volume change to be permitted before a
          mesh rezone is performed (Default = 0.10)
SDTTOL = Relative change in estimated stable time step to
          be permitted before a mesh rezone is performed
          (Default = 0.80)
HGDAMP = Hourglass damping fraction (Default = 0.3)

Notes :

(1) - Although automatic time-stepping is used in the
      numerical solution, high velocity problems often
      require a finer step than the computed stability
      limit. A small DTFRAC will be required when the
      velocities present in the problem approach local
      wave speeds in the material.

(2) - Rezoning will be performed if EITHER the volume
      change or time step change tolerance is exceeded.
      Rezoning is minimized when VOLTOL=1.0 and
      SDTTOL=0.0. As VOLTOL approaches 0 and/or DSTTOL
      approaches 1, rezoning will be performed more
      frequently (i.e., for less extreme distortions).

(3) - By default, no hourglass damping is used in the
      solution. When hourglass damping is used, values
      of HGDAMP in the range 0.2 - 0.3 are typical.
      Note that the use of hourglass damping decreases
      the allowable time step by a mutiplicative factor
      of SQRT(1.-HGDAMP**2)-HGDAMP; in fact, for values
      of HGDAMP above 0.7071, the allowable step size
      becomes zero.

```
--------------------------------
P R O P E R T I E S   Input Block
--------------------------------
```

Header : PROP
End   : (1 Record)

Record Format #1 :

```
. . . . . .10 . . . . . .20 . . . . . .30 . . . . .40 . . . .50 . . . .60
         RHO           G           KL           KQ          SY          ET
```

Input Variables :

RHO   = Reference value of material density (pressure=0)
G   = Elastic shear modulus
KL   = Linearized bulk modulus
KQ   = Quadratic pressure coefficient
SY   = Yield stress
ET   = Secondary (plastic) tangent modulus

Notes :

(1) - The material is assumed to obey an isotropic, hypoelastic
      constitutive law, in which the co-rotational rate of the
      Cauchy stress, denoted by $\overset{\triangledown}{\sigma}$, is related to the rate of
      deformation $\underline{d}$ by

$$\overset{\triangledown}{\sigma} = (K - 2G/3)\ I\ tr(d)\ +\ 2G\ d$$

The Jaumann stress rate $\overset{\triangledown}{\sigma}$ is related to the material time
derivative of σ through the definition

$$\overset{\triangledown}{\sigma} \ = \ d\sigma/dt \ - \ w\sigma \ + \ \sigma w$$

in which w is the spin tensor. The instantaneous bulk
modulus K is determined as described in Note (2) below.

(2)   The equation of state governing pressure-volume behavior
of the material is

$$p = K \ln (\rho/\rho_O)$$

where p is the hydrostatic pressure (positive in
compression). The instantaneous bulk modulus is defined
by

$$K = K_L \qquad\qquad \text{(tension, } \rho/\rho_O < 1)$$

$$K = K_L + \left(\frac{\rho}{\rho_O}-1\right)^2 K_Q \quad \text{(compression, } \rho/\rho_O > 1)$$

Therefore, the bulk behavior in compression exhibits a
"stiffening" effect which does not occur in tension.

(3)   When the effective stress in an element reaches the yield
value, the constitutive relationship used is of the
elastic-plastic-hydrodynamic type. Thus, material which
deforms permanently at very low stress can be simulated
by specifying a low value of SY.

90

```
-------------------------------------------------------
R E F E R E N C E   S T A T I O N S   Input Block
-------------------------------------------------------
```

Header : REFE
End   : --

Record Format #1 :

```
. . 5 . .10 . .15
   NX    NY    NZ
```

Record Format #2 : (repeat as needed to define NX reference
                    stations, then NY, then NZ)

```
. . . . .10 . . . . .20 . . .30 . . .40 . . ... . . .80
   REF(1)     REF(2)   REF(3)   REF(4)        ...   REF(8)
   REF(9)     REF(10)  REF(11)  REF(12)       ...   (etc.)
```

Input Variables :

NX/Y/Z = Number of rezone reference stations in the X, Y, and
         Z directions, respectively
REF(i) = i-th reference station in a particular direction. A
         new input line should be started for each of the X, Y,
         and Z directions. Values are entered 8 per line, in
         ascending order, on as many lines as necessary.

Notes:

  (1) - Presently, rezoning algorithm II (subsection 2.6.3) is
        used exclusively for rezoning. With this algorithm, the
        only effect of the reference station input is to establish
        constant plotting limits for the SBI solution, using the
        lowest and highest coordinate values in each direction.

Therefore, NX, NY, and NZ should each normally be set to
2. Specify X, Y, Z reference station values (2 each)
which define the limits for plotting throughout the
solution.

```
------------------------------------------------
R I G I D   W A L L   B . C .   Input Block
------------------------------------------------
```

Header : RIGI

End   : Blank line

Record Format #1 : (repeat as necessary to define all rigid
                    wall b.c.'s for the model)


```
. . .10 . . .20 . . .30 . . .40 . . .50 . . .60 . . .70 . . .80
     A       B       C       D       E       F       G       H
```


Input Variables :

A,B,... = Coefficients for a single rigid-wall boundary con-
          dition. The constraint defined by coefficients A,
          B, ... , H has the form:

          Ax + By + Cz - (D + EsinFt + GcosHt) > 0

          For example, to define the condition x>2, let A=1,
          B=C=0, D=2, and E=F=G=H=0. To define a constraint
          y<7, let A=C=E=F=G=H=0, B=-1 and D=-7. The E,F,G,
          and H terms permit the specification of time-vary-
          ing conditions, such as x > 2 sin 4t (A=1, E=2 and
          F=4).

```

```

S Y M M E T R Y   B . C . S   Input Block

```

Header : SYMM
End    : Blank line

Record Format #1 :

```
. . . 5 . . .10 . . .15 . . .20 . . .25 . . .30
   IDIR            VALUE  IFLAG1         VELOC1
```

```
. . .35 . . .40 . . .45 . . .50 . . .55 . . .60
 IFLAG2          VELOC2  IFLAG3         VELOC3
```

Input Variables :

IDIR   = Orientation of plane on which boundary condition is
         to be imposed (e.g., IDIR=1 for an X=constant plane).
VALUE = Position of plane (value of X(IDIR)).
IFLAGi = Switch for direction "i" for this boundary condition.
         If IFLAGi = 0, no constraints are imposed on the i-th
         component of velocity. When IFLAGi > 0, the velocity
         in the i-th coordinate direction is VELOCi.
VELOCi = Prescribed velocity for coordinate direction "i". The
         value of VELOCi is ignored if IFLAGi = 0.

Notes :

  (1) - The boundary condition defined by a typical input line
        could be stated as: "on the plane X(IDIR) = VALUE, for
        any direction i=1,2,3 for which IFLAGi > 0, the veloc-
        ity is equal to VELOCi".

  (2) - This constraint type may be used to define boundary
        conditions other than symmetry (although this is its
        most common use).

94

```
----------------------
T I T L E   Input Block
----------------------
```

Header : TITL
End   : (1 Record)

Record Format #1 :


```
. . .10 . . .20 . . .30 . . .40 . . ... . . .80
```
TITLE (up to 80 characters)

Input Variables :

TITLE = Any alphanumeric string, up to 80 characters long,
         to be used as a page header in the problem output.

5.4 MODELING GUIDELINES

Finite element modeling of a fully coupled soft body impact problem is no more difficult than modeling of more conventional structural dynamics problems. However, the radically different methods of solution employed in the two meshes and the techniques used to establish the impact interface require that certain guidelines be followed if accurate results are to be obtained. This section is devoted to the discussion of these specialized modeling requirements.

First, we wish to re-emphasize that the finite elements and the solution methodology used in the two finite element meshes are quite different. In the structure mesh, the finite elements are usually of higher order (quadratic), so that fewer elements are needed for high accuracy. Moreover, the structural response is computed using implicit methods which involve large matrix manipulations. This means that nodal numbering has an important effect upon the efficiency of the solution, and measures should be taken during model preparation to control the matrix bandwidth. The soft body mesh is composed of low-order elements, and larger numbers of elements are typical for realistic problems; however, the solution in this mesh is explicit (using small time steps and no matrix manipulations), and numbering of the mesh has absolutely no effect upon solution efficiency.

Boundary conditions in the soft body mesh are a potential source of solution error, since they are applied in a completely different manner from those in the structural mesh. Situations in which extreme care is necessary usually arise when symmetry is used to reduce the size of a problem. When symmetry conditions must be imposed in the soft body mesh, especially in the vicinity of an impact site, the following procedures are suggested:

- apply BOTH symmetry (SYMM) and rigid-wall (RIGI) boundary conditions along the symmetry edges of the soft-body mesh; and

● create additional interface elements which cross the symmetry boundaries in the vicinity of an impact.

The first measure is desirable because boundary conditions in the soft body mesh are applied not to specific nodes, but to all nodes which instantaneously lie on a specified surface. The RIGI (rigid-wall) conditions prevent nodes from crossing the symmetry plane, a situation which may occur when imposing the interface conditions in a contact zone; the SYMM (symmetry) conditions impose the desired zero-velocity conditions. If this procedure is not followed, the following sequence of events is possible:

(a) a node "penetrates" the structure mesh, and must be moved in order to enforce compatibility of the two meshes;

(b) when the node is moved (always normal to the surface of the structure mesh), it is pushed off the plane of symmetry, usually to the side opposite the model;

(c) the desired symmetry conditions are not imposed, since the node no longer lies on the symmetry plane; and

(d) in some cases, the subsequent motion of the node is such that it may actually pass the edge of the structure mesh.

The addition of a RIGId wall condition causes the node in question to be moved back to the symmetry plane immediately following step (b) above, and both the symmetry and contact boundary conditions will be satisfied.

The second modeling device mentioned above in connection with symmetry surface modeling involves the addition of interface elements which actually cross the symmetry plane. This measure ensures that the phenomenon mentioned in (d) above does not occur in pathological situations. The tolerances used in determining contact are necessarily very small, since making them larger will result in certain nodes applying the same forces twice to two or

97

more neighboring elements. In some instances, these small tolerances may cause true contact conditions on symmetry edges to be skipped. The suggested procedure is to create an additional "layer" of nodes in the structure mesh, with their motion tied to the motion of the original structural nodes by linear constraint equations. The extra layer of nodes then allows an additional boundary "layer" of interface elements to be specified, through which the soft-body mesh may not pass.

A final consideration involves controlling the solution time step in the soft body mesh in moderate to high velocity problems. Normally, the time step in this mesh is selected automatically, based on estimates of the maximum wave speed in the body. When impact conditions are specified, however, nodes in the soft body mesh must be adjusted to eliminate any "penetration" of the structure mesh, with velocities and accelerations on the impact surface being adjusted accordingly. When the product of the relative velocity in the two meshes and the time step in the soft body mesh is larger than the soft body mesh spacing, these nodal coordinate corrections may produce elements with negative volume or, at least, severe distortion within a single time step. In such situations, it is appropriate to specify a maximum time step for the soft body mesh to avoid problems of this type. A reasonable estimate of the maximum time step can be obtained by comparing the initial (usually uniform) velocity of the soft body mesh with the smallest mesh division in the same direction; a limiting time step of about 10-20% of the mesh spacing divided by the initial velocity is usually adequate.

5.5   ANALYSIS RESTART PROCEDURES FOR COUPLED SOLUTION

Full restart capabilities are available for the coupled solution program. Restart checkpoints are created automatically whenever a coupled solution is selected; saving the restart data simply requires that the restart data file be saved on disk when a segment of the analysis is complete. This "new" restart file is called "HYDNEW" in all machine versions of the code; default file extensions for FORTRAN data files are used on machines such as the VAX-11/780.

Restart of an interrupted solution is requested whenever the MAGNA "RESTART" option is invoked. Restart data is read from the existing file "HYDOLD" (as with HYDNEW, default file extensions are used where applicable). On CDC and CRAY computers, the HYDOLD file should be copied from permanent file storage to a local file in the input job stream.

Occasionally, it may be convenient to restart a coupled solution as a normal nonlinear dynamic analysis, omitting the hydrodynamic mesh entirely. An example is a hydrodynamic impact case in which the impact loading phase is complete, and the two meshes have separated; analysis of the post-impact vibration of the structural model does not require the use of the coupled solution option. To accomplish such a restart, simply restart the structural analysis as a normal nonlinear dynamic problem (i.e., changing IOPT(18) to zero). The files FSDATA and HYDOLD need not be supplied as input.

Note that the opposite case (restarting a normal analysis as a coupled one) is presently not permitted. Once a solution (or a segment of a solution) has been performed without the coupling option, it may be restarted only as an uncoupled analysis.

SECTION 6
SUMMARY AND CONCLUSIONS

The soft body impact analysis described in this report represents an attempt to treat this important class of problems without adopting heuristic or highly idealized models of impact loading. As such, it will find application in situations where impact loading and the resulting response are highly coupled and cannot be determined independently. Examples include bird impacts on aircraft transparencies and foreign object ingestion by air-breathing propulsion systems.

The analytical framework developed in this report is general in scope, and can be enhanced as experience in analyzing realistic problems accumulates. Thus far, our experience in simulating real soft-body impacts is limited, and correlation of analytical and experimental results has been impeded by a lack of mechanical properties data. However, several of the modeling exercises performed thus far have provided useful knowledge of a pragmatic nature. In particular, the following observations can be made based upon our present experience:

o with a fully three-dimensional model based on "primitive" variables, a slight amount of shear stiffness (or fluid viscosity) is always necessary for a stable solution which is free from spurious modes of deformation;

o the anti-hourglassing technique suggested in Reference [21] is quite reliable, and moderate hourglass damping ratios (0.2-0.3) are virtually always adequate;

o an elastic-plastic-failure model is somewhat easier to work with in soft-body impact problems than a Newtonian fluid model, since the fluid model parameters are difficult to obtain and since the physical situation normally involves at least some elastic behavior;

100

o the use of a nonlinear model for bulk behavior of the soft body material can produce predicted results which are radically different (and usually improved) from a linear pressure-volume model;

o the application of symmetry boundary conditions in combination with impact constraints requires careful modeling to maintain correct behavior of the hydrodynamic mesh (subsection 5.4);

o time steps in the soft body mesh can be computed quite accurately from the instantaneous properties of individual finite elements, permitting continuous and automatic adjustment of the solution step size (subsection 2.4);

o modification of the computed time step for the soft body mesh normally can be limited to specifying a maximum time step, when the impact velocity is large enough that some elements might be compressed by their entire length within a single time step upon impact (subsection 5.4);

o force information obtained from an impact solution may become inaccurate when too few steps are used in the structural mesh, since peak forces which occur early in the impact event may build and decay within a single time step (Section 3).

The next important step toward the routine analysis of practical problems in soft body impact is the validation of the present methodology using experimental data from impact tests on deformable structures. Full scale impact test results for transparency birdstrikes exist [22], and can be used for the validation. The collection of mechanical properties data for both real and substitute bird materials in support of this validation activity is recommended as well.

REFERENCES

1. B. D. Nichols and C. W. Hirt, "Numerical Simulation of Hydrodynamic Impact Loads on Cylinders," EPRI NP-824, Interim Report, June, 1978.

2. T. Belytschko and R. Mullen, "Two-Dimensional Fluid-Structure Impact Computations," ASME Paper 80-C2/PVP-139, Century 2 Pressure Vessels and Piping Conference, San Francisco, California, August, 1980.

3. O. C. Zienkiewicz and P. Bettess, "Fluid-Structure Dynamic Interaction and Wave Forces - An Introduction to Numerical Treatment," International Journal for Numerical Methods in Engineering, Vol. 13, pp. 1-16, 1978.

4. T. Belytschko, J. M. Kennedy, and D. F. Schoeberle, "Quasi-Eulerian Finite Element Formulation for Fluid-Structure Interaction," Trans. ASME, Journal of Pressure Vessel Technology, Vol. 102, No. 1, pp. 62-69, 1980.

5. R. F. Kulak, "A Three-Dimensional Finite Element Formulation for Fluid-Structure Interaction," Proceedings of the 5th Conference on Structural Mechanics in Reactor Technology (SMiRT), Berlin, Germany, August, 1979.

6. J. O. Hallquist, "Preliminary User's Manuals for DYNA3D and DYNAP (Nonlinear Dynamic Analysis of Solids in Three Dimensions)," University of California, Lawrence Livermore Laboratory, Report UCID-17268, Rev. 1, 1979.

7. G. R. Johnson, D. D. Colby, and D. J. Vavrick, "Further Development of the EPIC-3 Computer Program for Three-Dimensional Analysis of Intense Impulsive Loading," Air Force Armament Laboratory, AFATL-TR-78-81, July, 1978.

8. R. E. Durrett and D. A. Matuska, "The HULL Code. Finite Difference Solution to the Equations of Continuum Mechanics," Air Force Armament Laboratory, AFATL-TR-78-81, July, 1978.

9. J. J. Engblom, "Coupled Fluid/Structure Response Predictions for Soft Body Impact of Airfoil Configurations," in J. R. Vinson (editor), Emerging Technologies in Aerospace Structures, Design, Structural Dynamics and Materials, ASME Century 2 Publication, pp. 209-223, 1980.

10. R. L. Peterson and J. P. Barber, "Bird Impact Forces in Aircraft Windshield Design," AFFDL-TR-75-150, Flight Dynamics Laboratory, Wright-Patterson Air Force Base, Ohio, March, 1976.

11. B. S. West and R. A. Brockman, "Evaluation of Bird Load Models for Dynamic Analysis of Aircraft Transparencies," AFWAL-TR-80-3092, Flight Dynamics Laboratory, Air Force Wright Aeronautical Laboratories, Wright-Patterson Air Force Base, Ohio, August, 1980.

12. O. C. Zienkiewicz, The Finite Element Method, McGraw-Hill Book Co. (UK) Ltd., 1977.

13. _____, "ABAQUS -- Structural and Heat Transfer Analysis Program," Hibbitt, Karlsson and Sorensen, Inc., Providence, Rhode Island.

14. K. J. Bathe, "ADINA -- A Finite Element Program for Automatic Dynamic Incremental Nonlinear Analysis," Report No. 82448-1, Acoustics and Vibration Laboratory, Department of Mechanical Engineering, Massachusetts Institute of Technology, Cambridge, Massachusetts, Rev. December, 1978.

15. G. J. DeSalvo and J. A. Swanson, "ANSYS User's Manual," Swanson Analysis Systems, Inc., Houston, Pennsylvania, July, 1979.

16. R. A. Brockman, "MAGNA (Materially And Geometrically Nonlinear Analysis). Part I - Finite Element Analysis Manual," AFWAL-TR-82-3098, Part I, Flight Dynamics Laboratory, Air Force Wright Aeronautical Laboratories, Wright-Patterson Air Force Base, Ohio, December, 1982. Also UDR-TR-82-111, University of Dayton Research Institute, Dayton, Ohio, December, 1982.

17. _____, "MARC User Information Manual", Volumes A-G, MARC Analysis Research Corporation, Palo Alto, California.

18. R. A. Brockman, "Numerical Solution of Large Deformation Problems involving Surface Contact and Impact," in J. N. Reddy (editor), Penalty-Finite Elements in Mechanics, ASME, Applied Mechanics Division, AMD Vol. 51, pp. 123-136, 1982.

19. T. Belytschko, H. J. Yen, and R. Mullen, "Mixed Methods for Time Integration," Computer Methods in Applied Mechanics and Engineering, Vol. 17/18, pp.259-275, 1979.

20. K. J. Bathe and E. L. Wilson, <u>Numerical Methods in Finite Element Analysis</u>, Prentice-Hall Co., Englewood Cliffs, New Jersey, 1975.

21. D. P. Flanagan and T. Belytschko, "A Uniform Strain Hexahedron and Quadrilateral with Orthogonal Hourglass Control," <u>International Journal for Numerical Methods in Engineering</u>, Vol. 17, pp. 679-706, 1981.

22. R. M. Watt, "AFFDL F-16 Canopy Bird Impact Test (Final Checked Data)," Data Package, Project V415-18A, von Karman Gas Dynamics Facility, Arnold Engineering Development Center, Arnold Air Force Station, Tennessee, June, 1977.

23. J. P. Barber, H. R. Taylor, and J. S. Wilbeck, "Bird Impact Forces and Pressures on Rigid and Compliant Targets," AFFDL-TR-77-60, Air Force Flight Dynamics Laboratory, Wright-Patterson Air Force Base, Ohio, May, 1978.